Contents

Preface

Introduction

This text was written for an introductory course in fluid mechanics. Our approach to the subject, as in all previous editions, emphasizes the physical concepts of fluid mechanics and methods of analysis that begin from basic principles. The primary objective of this text is to help users develop an orderly approach to problem solving. Thus we always start from governing equations, state assumptions clearly, and try to relate mathematical results to corresponding physical behavior. We continue to emphasize the use of control volumes to maintain a practical problem-solving approach that is also theoretically inclusive.

Proven Problem-Solving Methodology

The Fox-McDonald-Pritchard solution methodology used in this text is illustrated in numerous Examples in each chapter. Solutions presented in the Examples have been prepared to illustrate good solution technique and to explain difficult points of theory. Examples are set apart in format from the text so that they are easy to identify and follow. Additional important information about the text and our procedures is given in the "Note to Student" in Section 1.1 of the printed text. We urge you to study this section carefully and to integrate the suggested procedures into your problem-solving and results-presentation approaches.

SI and English Units

SI units are used in about 70 percent of both Example and end-of-chapter problems. English Engineering units are retained in the remaining problems to provide experience with this traditional system and to highlight conversions among unit systems that may be derived from fundamentals.

Goals and Advantages of Using This Text

Complete explanations presented in the text, together with numerous detailed Examples, make this book understandable for students, freeing the instructor to depart from conventional lecture teaching methods. Classroom time can be used to bring in outside material, expand on special topics (such as non-Newtonian flow, boundary-layer flow, lift and drag, or experimental methods), solve example problems, or explain difficult points of assigned homework problems. In addition, the 51 Example *Excel* workbooks are useful for presenting a variety of fluid mechanics phenomena, especially the effects produced when varying input parameters. Thus each class period can be used in the manner most appropriate to meet student needs.

When students finish the fluid mechanics course, we expect them to be able to apply the governing equations to a variety of problems, including those they have not encountered previously. We particularly emphasize physical concepts throughout to help students model the variety of phenomena that occur in real fluid flow situations. Although we collect, for convenience, useful equations at the end of most chapters, we stress that our philosophy is to minimize the use of so-called magic formulas and emphasize the systematic and fundamental approach to problem solving. By following this format, we believe students develop confidence in their ability to apply the material and to find that they can reason out solutions to rather challenging problems.

The book is well suited for independent study by students or practicing engineers. Its readability and clear examples help build confidence. Answer to Selected Problems are included, so students may check their own work.

Topical Coverage

The material has been selected carefully to include a broad range of topics suitable for a one- or two-semester course at the junior or senior level. We assume a background in rigid-body dynamics and mathematics through differential equations. A background in thermodynamics is desirable for studying compressible flow.

More advanced material, not typically covered in a first course, has been moved to the Web site (these sections are identified in the Table of Contents as being on the Web site). Advanced material is available to interested users of the book; available online, it does not interrupt the topic flow of the printed text.

Material in the printed text has been organized into broad topic areas:

- Introductory concepts, scope of fluid mechanics, and fluid statics (Chapters 1, 2, and 3)
- Development and application of control volume forms of basic equations (Chapter 4)
- Development and application of differential forms of basic equations (Chapters 5 and 6)
- Dimensional analysis and correlation of experimental data (Chapter 7)
- Applications for internal viscous incompressible flows (Chapter 8)
- Applications for external viscous incompressible flows (Chapter 9)
- Analysis of fluid machinery and system applications (Chapter 10)
- Analysis and applications of open-channel flows (Chapter 11)
- Analysis and applications of one- and two-dimensional compressible flows (Chapters 12 and 13)

Chapter 4 deals with analysis using both finite and differential control volumes. The Bernoulli equation is derived (in an optional subsection of Section 4.4) as an example

application of the basic equations to a differential control volume. Being able to use the Bernoulli equation in Chapter 4 allows us to include more challenging problems dealing with the momentum equation for finite control volumes.

Another derivation of the Bernoulli equation is presented in Chapter 6, where it is obtained by integrating Euler's equation along a streamline. If an instructor chooses to delay introducing the Bernoulli equation, the challenging problems from Chapter 4 may be assigned during study of Chapter 6.

Text Features

This edition incorporates a number of useful features:

- *Examples*: Fifty-one of the Examples include *Excel* workbooks, available online at the text Web site, making them useful for what-if analyses by students or by the instructor during class.
- *Case Studies*: Every chapter begins with a *Case Studies in Energy and the Environment*, each describing an interesting application of fluid mechanics in the area of renewable energy or of improving machine efficiencies. We have also retained from the previous edition chapter-specific *Case Studies*, which are now located at the end of chapters. These explore unusual or intriguing applications of fluid mechanics in a number of areas.
- *Chapter Summary and Useful Equations*: At the end of most chapters we collect for the student's convenience the most used or most significant equations of the chapter. Although this is a convenience, we cannot stress enough the need for the student to ensure an understanding of the derivation and limitations of each equation before its use!
- *Design Problems*: Where appropriate, we have provided open-ended design problems in place of traditional laboratory experiments. For those who do not have complete laboratory facilities, students could be assigned to work in teams to solve these problems. Design problems encourage students to spend more time exploring applications of fluid mechanics principles to the design of devices and systems. As in the previous edition, design problems are included with the end-of-chapter problems
- *Open-Ended Problems*: We have included many open-ended problems. Some are thought-provoking questions intended to test understanding of fundamental concepts, and some require creative thought, synthesis, and/or narrative discussion. We hope these problems will help instructors to encourage their students to think and work in more dynamic ways, as well as to inspire each instructor to develop and use more open-ended problems.
- *End-of-Chapter Problems*: Problems in each chapter are arranged by topic, and within each topic they generally increase in complexity or difficulty. This makes it easy for the instructor to assign homework problems at the appropriate difficulty level for each section of the book. For convenience, problems are now grouped according to the chapter section headings.

New to This Edition

This edition incorporates a number of significant changes:

- *Case Studies in Energy and the Environment*: At the beginning of each chapter is a new case study. With these case studies we hope to provide a survey of the most interesting and novel applications of fluid mechanics with the goal of generating

increasing amounts of the world's energy needs from renewable sources. The case studies are not chapter specific; that is, each one is not necessarily based on the material of the chapter in which it is presented. Instead, we hope these new case studies will serve as a stimulating narrative on the field of renewable energy for the reader and that they will provide material for classroom discussion. The case studies from the previous edition have been retained and relocated to the ends of chapters.

CLASSIC VIDEO

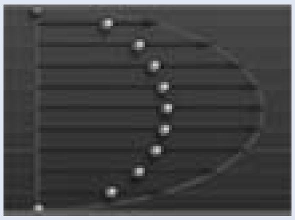
Classics!

- *Demonstration Videos*: The "classic" NCFMF videos (approximately 20 minutes each, with Professor Ascher Shapiro of MIT, a pioneer in the field of biomedical engineering and a leader in fluid mechanics research and education, explaining and demonstrating fluid mechanics concepts) referenced in the previous edition have all been retained and supplemented with additional new brief videos (approximately 30 seconds to 2 minutes each) from a variety of sources.

  Both the classic and new videos are intended to provide visual aids for many of the concepts covered in the text, and are available at www.wiley.com/college/pritchard.

VIDEO

New Videos!



- *CFD*: The section on basic concepts of computational fluid dynamics in Chapter 5 now includes material on using the spreadsheet for numerical analysis of simple one- and two-dimensional flows; it includes an introduction to the Euler method.
- *Fluid Machinery*: Chapter 10 has been restructured, presenting material for pumps and fans first, followed by a section on hydraulic turbines. Propellers and wind turbines are now presented together. The section on wind turbines now includes the analysis of vertical axis wind turbines (VAWTs) in additional depth. A section on compressible flow machines has also been added to familiarize students with the differences in evaluating performance of compressible versus incompressible flow machines. The data in Appendix D on pumps and fans has been updated to reflect new products and new means of presenting data.
- *Open-Channel Flow*: In this edition we have completely rewritten the material on open-channel flows. An innovation of this new material compared to similar texts is that we have treated "local" effects, including the hydraulic jump before considering uniform and gradually varying flows. This material provides a sufficient background on the topic for mechanical engineers and serves as an introduction for civil engineers.
- *Compressible Flow*: The material in Chapter 13 has been restructured so that normal shocks are discussed before Fanno and Rayleigh flows. This was done because many college fluid mechanics curriculums cover normal shocks but not Fanno or Rayleigh flows.
- *New Homework Problems*: The eighth edition includes 1705 end-of-chapter problems. Many problems have been combined and contain multiple parts. Most have been structured so that all parts need not be assigned at once, and almost 25 percent of subparts have been designed to explore what-if questions. New or modified for this edition are some 518 problems, some created by a panel of instructors and subject matter experts. End-of-chapter homework problems are now grouped according to text sections.

Resources for Instructors

The following resources are available to instructors who adopt this text. Visit the Web site at www.wiley.com/college/pritchard to register for a password.

- *Solutions Manual for Instructors*: The solutions manual for this edition contains a complete, detailed solution for all homework problems. Each solution is prepared in the same systematic way as the Example solutions in the printed text. Each solution

begins from governing equations, clearly states assumptions, reduces governing equations to computing equations, obtains an algebraic result, and finally substitutes numerical values to calculate a quantitative answer. Solutions may be reproduced for classroom or library use, eliminating the labor of problem solving for the instructor who adopts the text.

The *Solutions Manual* is available online after the text is adopted. Visit the instructor section of the text's Web site at www.wiley.com/college/pritchard to request access to the password-protected online *Solutions Manual*.

- *Problem Key*: A list of all problems that are renumbered from the seventh edition of this title, to the eighth edition. There is no change to the actual solution to each of these problems.
- *PowerPoint Lecture Slides*: Lecture slides have been developed by Philip Pritchard, outlining the concepts in the book, and including appropriate illustrations and equations.
- *Image Gallery*: Illustrations from the text in a format appropriate to include in lecture presentations.

Additional Resources

- *A Brief Review of Microsoft Excel*: Prepared by Philip Pritchard and included on the book Web site as Appendix H, this resource will coach students in setting up and solving fluid mechanics problems using *Excel* spreadsheets. Visit www.wiley.com/college/pritchard to access it.
- *Excel Files*: These *Excel* Files and add-ins are for use with specific Examples from the text.
- *Additional Text Topics*: PDF files for these topics/sections are available only on the Web site. These topics are highlighted in the text's table of contents and in the chapters as being available on the Web site.
- *Answers to Selected Problems*: Answers to odd-numbered problems are listed at the end of the book as a useful aid for student self-study.
- *Videos*: Many worthwhile videos are available on the book Web site to demonstrate and clarify the basic principles of fluid mechanics. When it is appropriate to view these videos to aid in understanding concepts or phenomena, an icon appears in the margin of the printed text; the Web site provides links to both classic and new videos, and these are also listed in Appendix C.

WileyPLUS

WileyPLUS is an innovative, research-based, online environment for effective teaching and learning.

What do students receive with WileyPLUS?

A Research-Based Design

WileyPLUS provides an online environment that integrates relevant resources, including the entire digital textbook, in an easy-to-navigate framework that helps students study more effectively.

- *WileyPLUS* adds structure by organizing textbook content into smaller, more manageable "chunks."
- Related media, examples, and sample practice items reinforce the learning objectives.
- Innovative features such as calendars, visual progress tracking and self-evaluation tools improve time management and strengthen areas of weakness.

One-on-One Engagement

With *WileyPLUS* for *Introduction to Fluid Mechanics*, eighth edition, students receive 24/7 access to resources that promote positive learning outcomes. Students engage with related examples (in various media) and sample practice items, including:

- Guided Online (GO) Tutorial problems
- Concept Questions
- Demonstration videos

Measurable Outcomes

Throughout each study session, students can assess their progress and gain immediate feedback. *WileyPLUS* provides precise reporting of strengths and weaknesses, as well as individualized quizzes, so that students are confident they are spending their time on the right things. With *WileyPLUS*, students always know the exact outcome of their efforts.

What do instructors receive with WileyPLUS?

WileyPLUS provides reliable, customizable resources that reinforce course goals inside and outside of the classroom as well as visibility into individual student progress. Pre-created materials and activities help instructors optimize their time:

Customizable Course Plan

WileyPLUS comes with a pre-created Course Plan designed by a subject matter expert uniquely for this course. Simple drag-and-drop tools make it easy to assign the course plan as-is or modify it to reflect your course syllabus.

Pre-created Activity Types Include

- Questions
- Readings and Resources

Course Materials and Assessment Content

- Lecture Notes PowerPoint Slides
- Image Gallery
- Gradable FE Exam sample Questions
- Question Assignments: Selected end-of-chapter problems coded algorithmically with hints, links to text, whiteboard/show work feature and instructor controlled problem solving help.

- Concept Question Assignments: Questions developed by Jay Martin and John Mitchell of the University of Wisconsin-Madison to assess students' conceptual understanding of fluid mechanics.

Gradebook

WileyPLUS provides instant access to reports on trends in class performance, student use of course materials, and progress towards learning objectives, helping inform decisions and drive classroom discussions.

WileyPLUS. Learn More. www.wileyplus.com.

Powered by proven technology and built on a foundation of cognitive research, *WileyPLUS* has enriched the education of millions of students, in over 20 countries around the world.

Acknowledgments

We recognize that no single approach can satisfy all needs, and we are grateful to the many students and faculty whose comments have helped us improve on earlier editions of this book.

We wish to express our thanks to the contributors and reviewers of the *WileyPLUS* course:

Darrell W. Pepper, University of Nevada, Las Vegas
Brian P. Sangeorzan, Oakland University
Asghar Esmaeeli, Southern Illinois University, Carbondale
Andrew Gerhart, Lawrence Technological University
John Mitchell, University of Wisconsin, Madison
David Benson, Kettering University
Donald Fenton, Kansas State University
Alison Griffin, University of Central Florida
John Leylegian, Manhattan College
Mark Cummings, University of Idaho

We would also like to thank Bud Homsy for his assistance in getting permission from Stanford University, as well as the University of California, Santa Barbara, to license many of the videos we are making available to adopters of this edition, and we thank Gordon McCreight for his assistance with this process as well.

The following individuals are thanked for their invaluable contributions in developing interesting new problems for several chapters:

Kenneth W. Miller, St. Cloud State University
Darrell W. Pepper, University of Nevada, Las Vegas
Shizhi Qian, Old Dominion University
Thomas Shepard, University of Minnesota

The eighth edition was carefully reviewed in whole or part by:

John Abbitt, University of Florida
Soyoung Stephen Cha, University of Illinois, Chicago
Kangping Chen, Arizona State University
W. Scott Crawford, Stanford University
Timothy J. Fry, University of Dayton
James W. Leach, North Carolina State University
Jed E. Marquart, Ohio Northern University
Hans Mayer, California Polytechnic State University, San Luis Obispo
Karl R. Nelson, Colorado School of Mines

Siva Parameswaran, Texas Tech University
Brian P. Sangeorzan, Oakland University
Brian Savilonis, Worcester Polytechnic Institute
Hayley H. Shen, Clarkson University

We are extremely grateful for their comments and suggestions.

Finally, for this edition, we are very deeply indebted to John Leylegian, of Manhattan College, for his major contributions to this edition. He restructured Chapter 10 (and revised Appendix D), and he made significant contributions to changes in all the other chapters. He also took major responsibility for revising, updating, or replacing end-of-chapter problems for half of the chapters, as well as generating the corresponding parts of the solution manual. His expertise was essential for the revisions to Chapter 10.

We look forward to continued interactions with these and other colleagues who use the book.

Professor Pritchard appreciates the unstinting support of his wife, Penelope, who is keenly aware of all the hours that went into the effort of preparing this edition.

We welcome suggestions and/or criticisms from interested users of this book.

Philip J. Pritchard
August 2010

Table G.1

SI Units and Prefixes[a]

SI Units	Quantity	Unit	SI Symbol	Formula
SI base units:	Length	meter	m	—
	Mass	kilogram	kg	—
	Time	second	s	—
	Temperature	kelvin	K	—
SI supplementary unit:	Plane angle	radian	rad	—
SI derived units:	Energy	joule	J	$N \cdot m$
	Force	newton	N	$kg \cdot m/s^2$
	Power	watt	W	J/s
	Pressure	pascal	Pa	N/m^2
	Work	joule	J	$N \cdot m$

SI prefixes	Multiplication Factor	Prefix	SI Symbol
	$1\ 000\ 000\ 000\ 000 = 10^{12}$	tera	T
	$1\ 000\ 000\ 000 = 10^{9}$	giga	G
	$1\ 000\ 000 = 10^{6}$	mega	M
	$1\ 000 = 10^{3}$	kilo	k
	$0.01 = 10^{-2}$	centi[b]	c
	$0.001 = 10^{-3}$	milli	m
	$0.000\ 001 = 10^{-6}$	micro	μ
	$0.000\ 000\ 001 = 10^{-9}$	nano	n
	$0.000\ 000\ 000\ 001 = 10^{-12}$	pico	p

[a]Source: ASTM Standard for Metric Practice E 380-97, 1997.

[b]To be avoided where possible.

Table G.2

Conversion Factors and Definitions

Fundamental Dimension	English Unit	Exact SI Value	Approximate SI Value
Length	1 in.	0.0254 m	—
Mass	1 lbm	0.453 592 37 kg	0.454 kg
Temperature	1°F	5/9 K	—

Definitions:

Acceleration of gravity:	$g = 9.8066$ m/s^2 (= 32.174 ft/s^2)
Energy:	Btu (British thermal unit) ≡ amount of energy required to raise the temperature of 1 lbm of water 1°F (1 Btu = 778.2 ft · lbf)
	kilocalorie ≡ amount of energy required to raise the temperature of 1 kg of water 1 K(1 kcal = 4187 J)
Length:	1 mile = 5280 ft; 1 nautical mile = 6076.1 ft = 1852 m (exact)
Power:	1 horsepower ≡ 550 ft · lbf/s
Pressure:	1 bar ≡ 10^5 Pa
Temperature:	degree Fahrenheit, $T_F = \frac{9}{5}T_C + 32$ (where T_C is degrees Celsius)
	degree Rankine, $T_R = T_F + 459.67$
	Kelvin, $T_K = T_C + 273.15$ (exact)
Viscosity:	1 Poise ≡ 0.1 kg/(m · s)
	1 Stoke ≡ 0.0001 m^2/s
Volume:	1 gal ≡ 231 in.^{3} (1 ft^3 = 7.48 gal)

Useful Conversion Factors:

Length:	1 ft = 0.3048 m	Power:	1 hp = 745.7 W
	1 in. = 25.4 mm		1 ft · lbf/s = 1.356 W
Mass:	1 lbm = 0.4536 kg		1 Btu/hr = 0.2931 W
	1 slug = 14.59 kg	Area	1 ft^2 = 0.0929 m^2
Force:	1 lbf = 4.448 N		1 acre = 4047 m^2
	1 kgf = 9.807 N	Volume:	1 ft^3 = 0.02832 m^3
Velocity:	1 ft/s = 0.3048 m/s		1 gal (US) = 0.003785 m^3
	1 ft/s = 15/22 mph		1 gal (US) = 3.785 L
	1 mph = 0.447 m/s	Volume flow rate:	1 ft^3/s = 0.02832 m^3/s
Pressure:	1 psi = 6.895 kPa		1 gpm = 6.309×10^{-5} m^3/s
	1 lbf/ft^2 = 47.88 Pa	Viscosity (dynamic)	1 lbf · s/ft^2 = 47.88 N · s/m^2
	1 atm = 101.3 kPa		1 g/(cm · s) = 0.1 N · s/m^2
	1 atm = 14.7 psi		1 Poise = 0.1 N · s/m^2
	1 in. Hg = 3.386 kPa	Viscosity (kinematic)	1 ft^2/s = 0.0929 m^2/s
	1 mm Hg = 133.3 Pa		1 Stoke = 0.0001 m^2/s
Energy:	1 Btu = 1.055 kJ		
	1 ft · lbf = 1.356 J		
	1 cal = 4.187 J		

1

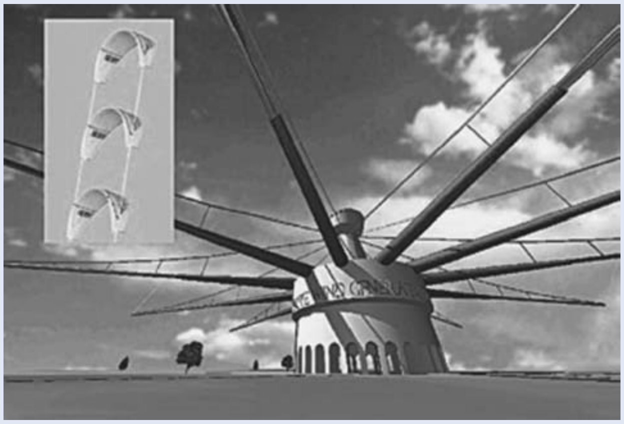
Introduction

Case Study in Energy and the Environment

Wind Power

At the beginning of each chapter we present a case study in the role of fluid mechanics in helping solve the energy crisis and in alleviating the environmental impact of our energy needs: the cases provide insight into the ongoing importance of the field of fluid mechanics. We have tried to present novel and original developments, not the kind of applications such as the ubiquitous wind farms. Please note that case studies represent a narrative; so each chapter's case study is not necessarily representative of the material in that chapter. Perhaps as a creative new engineer, you'll be able to create even better ways to extract renewable, nonpolluting forms of energy or invent something to make fluid-mechanics devices more efficient!

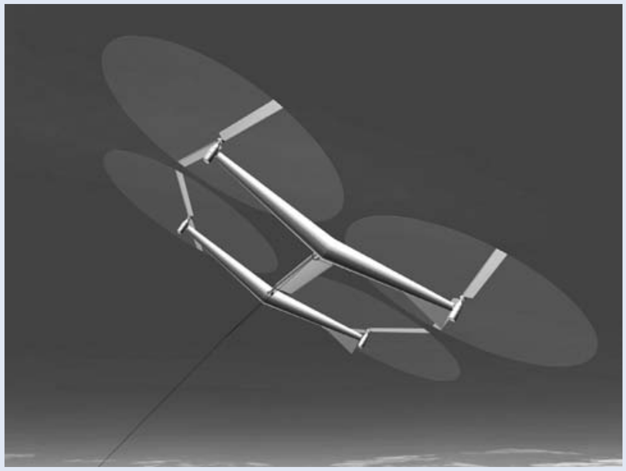
According to the July 16, 2009, edition of the *New York Times*, the global wind energy potential is much higher than previously estimated by both wind industry groups and government agencies. (Wind turbines are discussed in Chapter 10.) Using data from thousands of meteorological stations, the research indicates that the world's wind power potential is about 40 times greater than total current power consumption; previous studies had put that multiple at about seven times! In the lower 48 states, the potential from wind power is 16 times more



KiteGen's kites would fly at an altitude of about 1000 m and spin a power carousel on the ground. (Picture courtesy of Ben Shepard and Archer & Caldeira.)

than total electricity demand in the United States, the researchers suggested, again much higher than a 2008 Department of Energy study that projected wind could supply a fifth of all electricity in the country by 2030. The findings indicate the validity of the often made claim that "the United States is the Saudi Arabia of wind." The new estimate is based the idea of deploying 2.5- to 3-megawatt (MW) wind turbines in rural areas that are neither frozen nor forested and also on shallow offshore locations, and it includes a conservative 20 percent estimate for capacity factor, which is a measure of how much energy a given turbine actually produces. It has been estimated that the total power from the wind that could conceivably be extracted is about 72 terawatts (TW, 72×10^{12} watts). Bearing in mind that the *total* power consumption by all humans was about 16 TW (as of 2006), it is clear that wind energy could supply all the world's needs for the foreseeable future!

One reason for the new estimate is due to the increasingly common use of very large turbines that rise to almost 100 m, where wind speeds are greater. Previous wind studies were based on the use of 50- to 80-m turbines. In addition, to reach even higher elevations (and hence wind speed), two approaches have been proposed. In a recent paper, Professor Archer at



Sky Windpower's flying electric generators would fly at altitudes of about 10,000 m. (Picture courtesy of Ben Shepard and Archer & Caldeira.)

California State University and Professor Caldeira at the Carnegie Institution of Washington, Stanford, discussed some possibilities. One of these is a design of *KiteGen* (shown in the figure), consisting of tethered airfoils (kites) manipulated by a control unit and connected to a ground-based, carousel-shaped generator; the kites are maneuvered so that they drive the carousel, generating power, possibly as much as 100 MW. This approach would be best for the lowest few kilometers of the atmosphere. An approach using further increases in elevation is to generate electricity aloft and then transmit it to the surface with a tether. In the design proposed by *Sky Windpower*, four rotors are mounted on an airframe; the rotors both provide lift for the device and power electricity generation. The aircraft would lift themselves into place with supplied electricity to reach the desired altitude but would then generate up to 40 MW of power. Multiple arrays could be used for large-scale electricity generation. (Airfoils are discussed in Chapter 9.)

We shall examine some interesting developments in wind power in the *Case Studies in Energy and the Environment* in subsequent chapters.

We decided to title this textbook "Introduction to . . ." for the following reason: After studying the text, you will *not* be able to design the streamlining of a new car or an airplane, or design a new heart valve, or select the correct air extractors and ducting for a $100 million building; however, you *will* have developed a good understanding of the concepts behind all of these, and many other applications, and have made significant progress toward being ready to work on such state-of-the-art fluid mechanics projects.

To start toward this goal, in this chapter we cover some very basic topics: a case study, what fluid mechanics encompasses, the standard engineering definition of a fluid, and the basic equations and methods of analysis. Finally, we discuss some common engineering student pitfalls in areas such as unit systems and experimental analysis.

Note to Students 1.1

This is a student-oriented book: We believe it is quite comprehensive for an introductory text, and a student can successfully self-teach from it. However, most students will use the text in conjunction with one or two undergraduate courses. In either case, we recommend a thorough reading of the relevant chapters. In fact, a good approach is to read a chapter quickly once, then reread more carefully a second and even a third time, so that concepts develop a context and meaning. While students often find fluid mechanics quite challenging, we believe this approach, supplemented by your instructor's lectures that will hopefully amplify and expand upon the text material (if you are taking a course), will reveal fluid mechanics to be a fascinating and varied field of study.

Other sources of information on fluid mechanics are readily available. In addition to your professor, there are many other fluid mechanics texts and journals as well as the Internet (a recent Google search for "fluid mechanics" yielded 26.4 million links, including many with fluid mechanics calculators and animations!).

There are some prerequisites for reading this text. We assume you have already studied introductory thermodynamics, as well as statics, dynamics, and calculus; however, as needed, we will review some of this material.

It is our strong belief that one learns best by *doing*. This is true whether the subject under study is fluid mechanics, thermodynamics, or soccer. The fundamentals in any of these are few, and mastery of them comes through practice. *Thus it is extremely important that you solve problems*. The numerous problems included at the end of each chapter provide the opportunity to practice applying fundamentals to the solution of problems. Even though we provide for your convenience a summary of useful equations at the end of each chapter (except this one), you should avoid the temptation to adopt a so-called plug-and-chug approach to solving problems. Most of the problems are such that this approach simply will not work. In solving problems we strongly recommend that you proceed using the following logical steps:

1. State briefly and concisely (in your own words) the information given.
2. State the information to be found.
3. Draw a schematic of the system or control volume to be used in the analysis. Be sure to label the boundaries of the system or control volume and label appropriate coordinate directions.
4. Give the appropriate mathematical formulation of the *basic* laws that you consider necessary to solve the problem.
5. List the simplifying assumptions that you feel are appropriate in the problem.
6. Complete the analysis algebraically before substituting numerical values.
7. Substitute numerical values (using a consistent set of units) to obtain a numerical answer.
    a. Reference the source of values for any physical properties.
    b. Be sure the significant figures in the answer are consistent with the given data.
8. Check the answer and review the assumptions made in the solution to make sure they are reasonable.
9. Label the answer.

In your initial work this problem format may seem unnecessary and even long-winded. However, it is our experience that this approach to problem solving is ultimately the most efficient; it will also prepare you to be a successful professional, for which a major prerequisite is to be able to communicate information and the results of an analysis clearly and precisely. *This format is used in all Examples presented in this text*; answers to Examples are rounded to three significant figures.

Finally, *we strongly urge you to take advantage of the many Excel tools available for this book on the text Web site*, for use in solving problems. Many problems can be

solved much more quickly using these tools; occasional problems can *only* be solved with the tools or with an equivalent computer application.

1.2 Scope of Fluid Mechanics

As the name implies, fluid mechanics is the study of fluids at rest or in motion. It has traditionally been applied in such areas as the design of canal, levee, and dam systems; the design of pumps, compressors, and piping and ducting used in the water and air conditioning systems of homes and businesses, as well as the piping systems needed in chemical plants; the aerodynamics of automobiles and sub- and supersonic airplanes; and the development of many different flow measurement devices such as gas pump meters.

While these are still extremely important areas (witness, for example, the current emphasis on automobile streamlining and the levee failures in New Orleans in 2005), fluid mechanics is truly a “high-tech” or “hot” discipline, and many exciting areas have developed in the last quarter-century. Some examples include environmental and energy issues (e.g., containing oil slicks, large-scale wind turbines, energy generation from ocean waves, the aerodynamics of large buildings, and the fluid mechanics of the atmosphere and ocean and of phenomena such as tornadoes, hurricanes, and tsunamis); biomechanics (e.g., artificial hearts and valves and other organs such as the liver; understanding of the fluid mechanics of blood, synovial fluid in the joints, the respiratory system, the circulatory system, and the urinary system); sport (design of bicycles and bicycle helmets, skis, and sprinting and swimming clothing, and the aerodynamics of the golf, tennis, and soccer ball); “smart fluids” (e.g., in automobile suspension systems to optimize motion under all terrain conditions, military uniforms containing a fluid layer that is “thin” until combat, when it can be “stiffened” to give the soldier strength and protection, and fluid lenses with humanlike properties for use in cameras and cell phones); and microfluids (e.g., for extremely precise administration of medications).

These are just a small sampling of the newer areas of fluid mechanics. They illustrate how the discipline is still highly relevant, and increasingly diverse, even though it may be thousands of years old.

1.3 Definition of a Fluid

CLASSIC VIDEO

Deformation of Continuous Media.

We already have a common-sense idea of when we are working with a fluid, as opposed to a solid: Fluids tend to flow when we interact with them (e.g., when you stir your morning coffee); solids tend to deform or bend (e.g., when you type on a keyboard, the springs under the keys compress). Engineers need a more formal and precise definition of a fluid: A *fluid* is a substance that deforms continuously under the application of a shear (tangential) stress no matter how small the shear stress may be. Because the fluid motion continues under the application of a shear stress, we can also define a fluid as any substance that cannot sustain a shear stress when at rest.

Hence liquids and gases (or vapors) are the forms, or phases, that fluids can take. We wish to distinguish these phases from the solid phase of matter. We can see the difference between solid and fluid behavior in Fig. 1.1. If we place a specimen of either substance between two plates (Fig. 1.1*a*) and then apply a shearing force F, each will initially deform (Fig. 1.1*b*); however, whereas a solid will then be at rest (assuming the force is not large enough to go beyond its elastic limit), a fluid will *continue* to deform (Fig. 1.1*c*, Fig. 1.1*d*, etc) as long as the force is applied. Note that a fluid in contact with a solid surface does not slip—it has the same velocity as that surface because of the *no-slip* condition, an experimental fact.

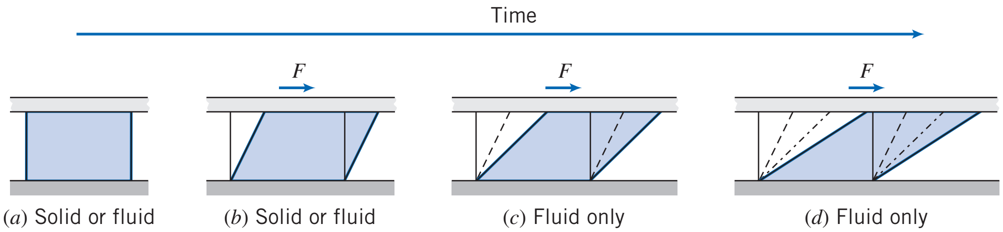


Fig. 1.1 Difference in behavior of a solid and a fluid due to a shear force.

CLASSIC VIDEO

Fundamentals—Boundary Layers.

The amount of deformation of the solid depends on the solid's modulus of rigidity G; in Chapter 2 we will learn that the *rate of deformation* of the fluid depends on the fluid's viscosity μ. We refer to solids as being *elastic* and fluids as being *viscous*. More informally, we say that solids exhibit "springiness." For example, when you drive over a pothole, the car bounces up and down due to the car suspension's metal coil springs compressing and expanding. On the other hand, fluids exhibit friction effects so that the suspension's shock absorbers (containing a fluid that is forced through a small opening as the car bounces) dissipate energy due to the fluid friction, which stops the bouncing after a few oscillations. If your shocks are "shot," the fluid they contained has leaked out so that there is almost no friction as the car bounces, and it bounces several times rather than quickly coming to rest. The idea that substances can be categorized as being either a solid or a liquid holds for most substances, but a number of substances exhibit both springiness and friction; they are *viscoelastic*. Many biological tissues are viscoelastic. For example, the synovial fluid in human knee joints lubricates those joints but also absorbs some of the shock occurring during walking or running. Note that the system of springs and shock absorbers comprising the car suspension is also viscoelastic, although the individual components are not. We will have more to say on this topic in Chapter 2.

Basic Equations 1.4

Analysis of any problem in fluid mechanics necessarily includes statement of the basic laws governing the fluid motion. The basic laws, which are applicable to any fluid, are:

1. The conservation of mass
2. Newton's second law of motion
3. The principle of angular momentum
4. The first law of thermodynamics
5. The second law of thermodynamics

Not all basic laws are always required to solve any one problem. On the other hand, in many problems it is necessary to bring into the analysis additional relations that describe the behavior of physical properties of fluids under given conditions.

For example, you probably recall studying properties of gases in basic physics or thermodynamics. The *ideal gas* equation of state

$$p = \rho R T \tag{1.1}$$

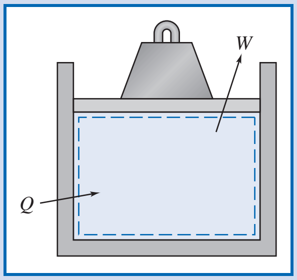
is a model that relates density to pressure and temperature for many gases under normal conditions. In Eq. 1.1, R is the gas constant. Values of R are given in Appendix A for several common gases; p and T in Eq. 1.1 are the absolute pressure and absolute temperature, respectively; ρ is density (mass per unit volume). Example 1.1 illustrates use of the ideal gas equation of state.

Example 1.1 FIRST LAW APPLICATION TO CLOSED SYSTEM

A piston-cylinder device contains 0.95 kg of oxygen initially at a temperature of 27°C and a pressure due to the weight of 150 kPa (abs). Heat is added to the gas until it reaches a temperature of 627°C. Determine the amount of heat added during the process.

Given: Piston-cylinder containing O_2, $m = 0.95$ kg.

$$T_1 = 27°\text{C} \qquad T_2 = 627°\text{C}$$



Find: $Q_{1\to 2}$.

Solution: $p = \text{constant} = 150$ kPa (abs)
We are dealing with a system, $m = 0.95$ kg.

Governing equation: First law for the system, $Q_{12} - W_{12} = E_2 - E_1$

Assumptions: (1) $E = U$, since the system is stationary.
(2) Ideal gas with constant specific heats.

Under the above assumptions,

$$E_2 - E_1 = U_2 - U_1 = m(u_2 - u_1) = mc_v(T_2 - T_1)$$

The work done during the process is moving boundary work

$$W_{12} = \int_{\forall_1}^{\forall_2} p\,d\forall = p(\forall_2 - \forall_1)$$

For an ideal gas, $p\forall = mRT$. Hence $W_{12} = mR(T_2 - T_1)$. Then from the first law equation,

$$Q_{12} = E_2 - E_1 + W_{12} = mc_v(T_2 - T_1) + mR(T_2 - T_1)$$
$$Q_{12} = m(T_2 - T_1)(c_v + R)$$
$$Q_{12} = mc_p(T_2 - T_1) \quad \{R = c_p - c_v\}$$

From the Appendix, Table A.6, for O_2, $c_p = 909.4$ J/(kg·K). Solving for Q_{12}, we obtain

$$Q_{12} = 0.95\,\text{kg} \times 909\,\frac{\text{J}}{\text{kg}\cdot\text{K}} \times 600\,\text{K} = 518\,\text{kJ} \longleftarrow \underline{Q_{12}}$$

This problem:
- ✓ Was solved using the nine logical steps discussed earlier.
- ✓ Reviewed use of the ideal gas equation and the first law of thermodynamics for a system.

It is obvious that the basic laws with which we shall deal are the same as those used in mechanics and thermodynamics. Our task will be to formulate these laws in suitable forms to solve fluid flow problems and to apply them to a wide variety of situations.

We must emphasize that there are, as we shall see, many apparently simple problems in fluid mechanics that cannot be solved analytically. In such cases we must resort to more complicated numerical solutions and/or results of experimental tests.

1.5 Methods of Analysis

The first step in solving a problem is to define the system that you are attempting to analyze. In basic mechanics, we made extensive use of the *free-body diagram*. We will

use a *system* or a *control volume*, depending on the problem being studied. These concepts are identical to the ones you used in thermodynamics (except you may have called them *closed system* and *open system*, respectively). We can use either one to get mathematical expressions for each of the basic laws. In thermodynamics they were mostly used to obtain expressions for conservation of mass and the first and second laws of thermodynamics; in our study of fluid mechanics, we will be most interested in conservation of mass and Newton's second law of motion. In thermodynamics our focus was energy; in fluid mechanics it will mainly be forces and motion. We must always be aware of whether we are using a system or a control volume approach because each leads to different mathematical expressions of these laws. At this point we review the definitions of systems and control volumes.

System and Control Volume

A *system* is defined as a fixed, identifiable quantity of mass; the system boundaries separate the system from the surroundings. The boundaries of the system may be fixed or movable; however, no mass crosses the system boundaries.

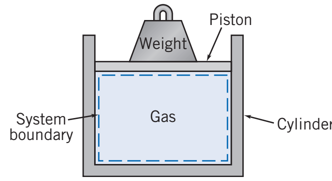


Fig. 1.2 Piston-cylinder assembly.

In the familiar piston-cylinder assembly from thermodynamics, Fig. 1.2, the gas in the cylinder is the system. If the gas is heated, the piston will lift the weight; the boundary of the system thus moves. Heat and work may cross the boundaries of the system, but the quantity of matter within the system boundaries remains fixed. No mass crosses the system boundaries.

In mechanics courses you used the free-body diagram (system approach) extensively. This was logical because you were dealing with an easily identifiable rigid body. However, in fluid mechanics we normally are concerned with the flow of fluids through devices such as compressors, turbines, pipelines, nozzles, and so on. In these cases it is difficult to focus attention on a fixed identifiable quantity of mass. It is much more convenient, for analysis, to focus attention on a volume in space through which the fluid flows. Consequently, we use the control volume approach.

A *control volume* is an arbitrary volume in space through which fluid flows. The geometric boundary of the control volume is called the control surface. The control surface may be real or imaginary; it may be at rest or in motion. Figure 1.3 shows flow through a pipe junction, with a control surface drawn on it. Note that some regions of the surface correspond to physical boundaries (the walls of the pipe) and others (at locations ①, ②, and ③) are parts of the surface that are imaginary (inlets or outlets). For the control volume defined by this surface, we could write equations for the basic laws and obtain results such as the flow rate at outlet ③ given the flow rates at inlet ①

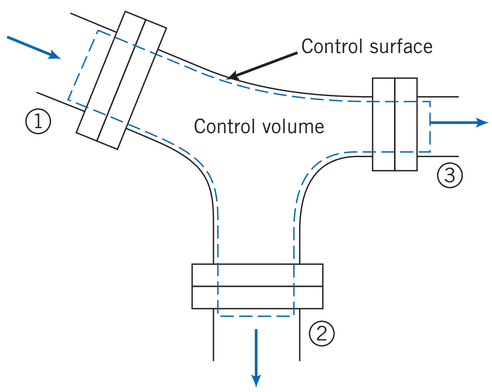
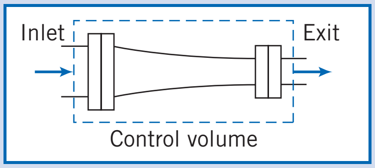


Fig. 1.3 Fluid flow through a pipe junction.

Example 1.2 MASS CONSERVATION APPLIED TO CONTROL VOLUME

A reducing water pipe section has an inlet diameter of 50 mm and exit diameter of 30 mm. If the steady inlet speed (averaged across the inlet area) is 2.5 m/s, find the exit speed.



Given: Pipe, inlet $D_i = 50$ mm, exit $D_e = 30$ mm.
Inlet speed, $V_i = 2.5$ m/s.

Find: Exit speed, V_e.

Solution:

Assumption: Water is incompressible (density $\rho =$ constant).

The physical law we use here is the conservation of mass, which you learned in thermodynamics when studying turbines, boilers, and so on. You may have seen mass flow at an inlet or outlet expressed as either $\dot{m} = VA/v$ or $\dot{m} = \rho VA$ where V, A, v, and ρ are the speed, area, specific volume, and density, respectively. We will use the density form of the equation.

Hence the mass flow is:

$$\dot{m} = \rho VA$$

Applying mass conservation, from our study of thermodynamics,

$$\rho V_i A_i = \rho V_e A_e$$

(Note: $\rho_i = \rho_e = \rho$ by our first assumption.)
(Note: Even though we are already familiar with this equation from thermodynamics, we will derive it in Chapter 4.)

Solving for V_e,

$$V_e = V_i \frac{A_i}{A_e} = V_i \frac{\pi D_i^2/4}{\pi D_e^2/4} = V_i \left(\frac{D_i}{D_e}\right)^2$$

$$V_e = 2.7 \frac{\text{m}}{\text{s}} \left(\frac{50}{30}\right)^2 = 7.5 \frac{\text{m}}{\text{s}} \longleftarrow V_e$$

This problem:
✓ Was solved using the nine logical steps.
✓ Demonstrated use of a control volume and the mass conservation law.

and outlet ② (similar to a problem we will analyze in Example 4.1 in Chapter 4), the force required to hold the junction in place, and so on. It is always important to take care in selecting a control volume, as the choice has a big effect on the mathematical form of the basic laws. We will illustrate the use of a control volume with an example.

Differential versus Integral Approach

The basic laws that we apply in our study of fluid mechanics can be formulated in terms of *infinitesimal* or *finite* systems and control volumes. As you might suspect, the equations will look different in the two cases. Both approaches are important in the study of fluid mechanics and both will be developed in the course of our work.

In the first case the resulting equations are differential equations. Solution of the differential equations of motion provides a means of determining the detailed behavior of the flow. An example might be the pressure distribution on a wing surface.

Frequently the information sought does not require a detailed knowledge of the flow. We often are interested in the gross behavior of a device; in such cases it is more appropriate to use integral formulations of the basic laws. An example might be the overall lift a wing produces. Integral formulations, using finite systems or control volumes, usually are easier to treat analytically. The basic laws of mechanics and thermodynamics, formulated in terms of finite systems, are the basis for deriving the control volume equations in Chapter 4.

Methods of Description

Mechanics deals almost exclusively with systems; you have made extensive use of the basic equations applied to a fixed, identifiable quantity of mass. On the other hand, attempting to analyze thermodynamic devices, you often found it necessary to use a control volume (open system) analysis. Clearly, the type of analysis depends on the problem.

Where it is easy to keep track of identifiable elements of mass (e.g., in particle mechanics), we use a method of description that follows the particle. This sometimes is referred to as the *Lagrangian* method of description.

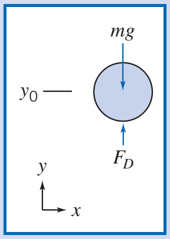
Consider, for example, the application of Newton's second law to a particle of fixed mass. Mathematically, we can write Newton's second law for a system of mass m as

$$\Sigma \vec{F} = m\vec{a} = m\frac{d\vec{V}}{dt} = m\frac{d^2\vec{r}}{dt^2} \tag{1.2}$$

In Eq. 1.2, $\Sigma \vec{F}$ is the sum of all external forces acting on the system, $\vec{a}$ is the acceleration of the center of mass of the system, $\vec{V}$ is the velocity of the center of mass of the system, and $\vec{r}$ is the position vector of the center of mass of the system relative to a fixed coordinate system.

Example 1.3 FREE FALL OF BALL IN AIR

The air resistance (drag force) on a 200 g ball in free flight is given by $F_D = 2 \times 10^{-4}\, V^2$, where F_D is in newtons and V is in meters per second. If the ball is dropped from rest 500 m above the ground, determine the speed at which it hits the ground. What percentage of the terminal speed is the result? (The *terminal speed* is the steady speed a falling body eventually attains.)



Given: Ball, $m = 0.2$ kg, released from rest at $y_0 = 500$ m.
Air resistance, $F_D = kV^2$, where $k = 2 \times 10^{-4}\ \text{N} \cdot \text{s}^2/\text{m}^2$.
Units: F_D(N), V(m/s).

Find: (a) Speed at which the ball hits the ground.
(b) Ratio of speed to terminal speed.

Solution:

Governing equation: $\Sigma \vec{F} = m\vec{a}$

Assumption: Neglect buoyancy force.

The motion of the ball is governed by the equation

$$\Sigma F_y = ma_y = m\frac{dV}{dt}$$

Since $V = V(y)$, we write $\Sigma F_y = m\dfrac{dV}{dy}\dfrac{dy}{dt} = mV\dfrac{dV}{dy}$ Then,

$$\Sigma F_y = F_D - mg = kV^2 - mg = mV\frac{dV}{dy}$$

Separating variables and integrating,

$$\int_{y_0}^{y} dy = \int_0^V \frac{mVdV}{kV^2 - mg}$$

$$y - y_0 = \left[\frac{m}{2k}\ln(kV^2 - mg)\right]_0^V = \frac{m}{2k}\ln\frac{kV^2 - mg}{-mg}$$

Taking antilogarithms, we obtain

$$kV^2 - mg = -mg\ e^{[(2k/m)(y-y_0)]}$$

Solving for V gives

$$V = \left\{\frac{mg}{k}\left(1 - e^{[(2k/m)(y-y_0)]}\right)\right\}^{1/2}$$

Substituting numerical values with $y = 0$ yields

$$V = \left\{0.2\,\text{kg} \times 9.81\frac{\text{m}}{\text{s}^2} \times \frac{\text{m}^2}{2\times10^{-4}\text{N}\cdot\text{s}^2} \times \frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}}\left(1 - e^{[2\times2\times10^{-4}/0.2(-500)]}\right)\right\}$$

$$V = 78.7\,\text{m/s} \longleftarrow V$$

At terminal speed, $a_y = 0$ and $\Sigma F_y = 0 = kV_t^2 - mg$.

Then, $V_t = \left[\dfrac{mg}{k}\right]^{1/2} = \left[0.2\,\text{kg} \times 9.81\,\dfrac{\text{m}}{\text{s}^2} \times \dfrac{\text{m}^2}{2\times10^{-4}\text{N}\cdot\text{s}^2} \times \dfrac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}}\right]^{1/2}$

$= 99.0$ m/s

The ratio of actual speed to terminal speed is

$$\frac{V}{V_t} = \frac{78.7}{99.0} = 0.795,\ \text{or } 79.5\% \longleftarrow \frac{V}{V_t}$$

This problem:
- ✓ Reviewed the methods used in particle mechanics.
- ✓ Introduced a variable aerodynamic drag force.

Try the *Excel* workbook for this Example for variations on this problem.

We could use this Lagrangian approach to analyze a fluid flow by assuming the fluid to be composed of a very large number of particles whose motion must be described. However, keeping track of the motion of each fluid particle would become a horrendous bookkeeping problem. Consequently, a particle description becomes unmanageable. Often we find it convenient to use a different type of description. Particularly with control volume analyses, it is convenient to use the field, or *Eulerian*, method of description, which focuses attention on the properties of a flow at a given point in space as a function of time. In the Eulerian method of description, the properties of a flow field are described as functions of space coordinates and time. We shall see in Chapter 2 that this method of description is a logical outgrowth of the assumption that fluids may be treated as continuous media.

Dimensions and Units 1.6

Engineering problems are solved to answer specific questions. It goes without saying that the answer must include units. In 1999, NASA's Mars Climate Observer crashed because the JPL engineers assumed that a measurement was in meters, but the supplying company's engineers had actually made the measurement in feet! Consequently, it is appropriate to present a brief review of dimensions and units. We say "review" because the topic is familiar from your earlier work in mechanics.

We refer to physical quantities such as length, time, mass, and temperature as *dimensions*. In terms of a particular system of dimensions, all measurable quantities are subdivided into two groups—*primary* quantities and *secondary* quantities. We refer to a small group of dimensions from which all others can be formed as primary quantities, for which we set up arbitrary scales of measure. Secondary quantities are those quantities whose dimensions are expressible in terms of the dimensions of the primary quantities.

Units are the arbitrary names (and magnitudes) assigned to the primary dimensions adopted as standards for measurement. For example, the primary dimension of length may be measured in units of meters, feet, yards, or miles. These units of length are related to each other through unit conversion factors (1 mile = 5280 feet = 1609 meters).

Systems of Dimensions

Any valid equation that relates physical quantities must be dimensionally homogeneous; each term in the equation must have the same dimensions. We recognize that Newton's second law ($\vec{F} \propto m\vec{a}$) relates the four dimensions, F, M, L, and t. Thus force and mass cannot both be selected as primary dimensions without introducing a constant of proportionality that has dimensions (and units).

Length and time are primary dimensions in all dimensional systems in common use. In some systems, mass is taken as a primary dimension. In others, force is selected as a primary dimension; a third system chooses both force and mass as primary dimensions. Thus we have three basic systems of dimensions, corresponding to the different ways of specifying the primary dimensions.

a. Mass $[M]$, length $[L]$, time $[t]$, temperature $[T]$
b. Force $[F]$, length $[L]$, time $[t]$, temperature $[T]$
c. Force $[F]$, mass $[M]$, length $[L]$, time $[t]$, temperature $[T]$

In system *a*, force $[F]$ is a secondary dimension and the constant of proportionality in Newton's second law is dimensionless. In system *b*, mass $[M]$ is a secondary dimension, and again the constant of proportionality in Newton's second law is dimensionless. In system *c*, both force $[F]$ and mass $[M]$ have been selected as primary dimensions. In this case the constant of proportionality, g_c (not to be confused with g, the acceleration of gravity!) in Newton's second law (written $\vec{F} = m\vec{a}/g_c$) is not dimensionless. The dimensions of g_c must in fact be $[ML/Ft^2]$ for the equation to be dimensionally homogeneous. The numerical value of the constant of proportionality depends on the units of measure chosen for each of the primary quantities.

Systems of Units

There is more than one way to select the unit of measure for each primary dimension. We shall present only the more common engineering systems of units for each of the basic systems of dimensions. Table 1.1 shows the basic units assigned to the primary dimensions for these systems. The units in parentheses are those assigned to that unit

Table 1.1

Common Unit Systems

System of Dimensions	Unit System	Force F	Mass M	Length L	Time t	Temperature T
a. MLtT	Système International d'Unités (SI)	(N)	kg	m	s	K
b. FLtT	British Gravitational (BG)	lbf	(slug)	ft	s	°R
c. FMLtT	English Engineering (EE)	lbf	lbm	ft	s	°R

system's secondary dimension. Following the table is a brief description of each of them.

a. MLtT

SI, which is the official abbreviation in all languages for the Système International d'Unités,[1] is an extension and refinement of the traditional metric system. More than 30 countries have declared it to be the only legally accepted system.

In the SI system of units, the unit of mass is the kilogram (kg), the unit of length is the meter (m), the unit of time is the second (s), and the unit of temperature is the kelvin (K). Force is a secondary dimension, and its unit, the newton (N), is defined from Newton's second law as

$$1\,\mathrm{N} \equiv 1\mathrm{kg}\cdot\mathrm{m/s^2}$$

In the Absolute Metric system of units, the unit of mass is the gram, the unit of length is the centimeter, the unit of time is the second, and the unit of temperature is the kelvin. Since force is a secondary dimension, the unit of force, the dyne, is defined in terms of Newton's second law as

$$1\,\mathrm{dyne} \equiv 1\mathrm{g}\cdot\mathrm{cm/s^2}$$

b. FLtT

CLASSIC VIDEO

Fluid Quantity and Flow.

In the British Gravitational system of units, the unit of force is the pound (lbf), the unit of length is the foot (ft), the unit of time is the second, and the unit of temperature is the degree Rankine (°R). Since mass is a secondary dimension, the unit of mass, the slug, is defined in terms of Newton's second law as

$$1\,\mathrm{slug} \equiv 1\,\mathrm{lbf}\cdot\mathrm{s^2/ft}$$

c. FMLtT

In the English Engineering system of units, the unit of force is the pound force (lbf), the unit of mass is the pound mass (lbm), the unit of length is the foot, the unit of time is the second, and the unit of temperature is the degree Rankine. Since both force and mass are chosen as primary dimensions, Newton's second law is written as

$$\vec{F} = \frac{m\vec{a}}{g_c}$$

[1] American Society for Testing and Materials, *ASTM Standard for Metric Practice*, E380-97. Conshohocken, PA: ASTM, 1997.

A force of one pound (1 lbf) is the force that gives a pound mass (1 lbm) an acceleration equal to the standard acceleration of gravity on Earth, 32.2 ft/s^2. From Newton's second law we see that

$$1\ \text{lbf} \equiv \frac{1\ \text{lbm} \times 32.2\ \text{ft/s}^2}{g_c}$$

or

$$g_c \equiv 32.2\ \text{ft} \cdot \text{lbm}/(\text{lbf} \cdot \text{s}^2)$$

The constant of proportionality, g_c, has both dimensions and units. The dimensions arose because we selected both force and mass as primary dimensions; the units (and the numerical value) are a consequence of our choices for the standards of measurement.

Since a force of 1 lbf accelerates 1 lbm at 32.2 ft/s^2, it would accelerate 32.2 lbm at 1 ft/s^2. A slug also is accelerated at 1 ft/s^2 by a force of 1 lbf. Therefore,

$$1\ \text{slug} \equiv 32.2\ \text{lbm}$$

Many textbooks and references use lb instead of lbf or lbm, leaving it up to the reader to determine from the context whether a force or mass is being referred to.

Preferred Systems of Units

In this text we shall use both the *SI* and the *British Gravitational* systems of units. In either case, the constant of proportionality in Newton's second law is dimensionless and has a value of unity. Consequently, Newton's second law is written as $\vec{F} = m\vec{a}$. In these systems, it follows that the gravitational force (the "weight"[2]) on an object of mass m is given by $W = mg$.

SI units and prefixes, together with other defined units and useful conversion factors, are summarized in Appendix G.

Example 1.4 USE OF UNITS

The label on a jar of peanut butter states its net weight is 510 g. Express its mass and weight in SI, BG, and EE units.

Given: Peanut butter "weight," $m = 510$ g.

Find: Mass and weight in SI, BG, and EE units.

Solution: This problem involves unit conversions and use of the equation relating weight and mass:

$$W = mg$$

The given "weight" is actually the mass because it is expressed in units of mass:

$$m_{SI} = 0.510\ \text{kg} \longleftarrow \underline{m_{SI}}$$

Using the conversions of Table G.2 (Appendix G),

$$m_{EE} = m_{SI}\left(\frac{1\ \text{lbm}}{0.454\ \text{kg}}\right) = 0.510\ \text{kg}\left(\frac{1\ \text{lbm}}{0.454\ \text{kg}}\right) = 1.12\ \text{lbm} \longleftarrow \underline{m_{EE}}$$

[2]Note that in the English Engineering system, the weight of an object is given by $W = mg/g_c$.

Using the fact that 1 slug = 32.2 lbm,

$$m_{\mathrm{BG}} = m_{\mathrm{EE}}\left(\frac{1\ \mathrm{slug}}{32.2\ \mathrm{lbm}}\right) = 1.12\ \mathrm{lbm}\left(\frac{1\ \mathrm{slug}}{32.2\ \mathrm{lbm}}\right)$$

$$= 0.0349\ \mathrm{slug} \longleftarrow m_{\mathrm{BG}}$$

To find the weight, we use

$$W = mg$$

In SI units, and using the definition of a newton,

$$W_{\mathrm{SI}} = 0.510\ \mathrm{kg} \times 9.81\ \frac{\mathrm{m}}{\mathrm{s}^2} = 5.00\left(\frac{\mathrm{kg}\cdot\mathrm{m}}{\mathrm{s}^2}\right)\left(\frac{\mathrm{N}}{\mathrm{kg}\cdot\mathrm{m/s}^2}\right)$$

$$= 5.00\ \mathrm{N} \longleftarrow W_{\mathrm{SI}}$$

In BG units, and using the definition of a slug,

$$W_{\mathrm{BG}} = 0.0349\ \mathrm{slug} \times 32.2\ \frac{\mathrm{ft}}{\mathrm{s}^2} = 1.12\frac{\mathrm{slug}\cdot\mathrm{ft}}{\mathrm{s}^2}$$

$$= 1.12\left(\frac{\mathrm{slug}\cdot\mathrm{ft}}{\mathrm{s}^2}\right)\left(\frac{\mathrm{s}^2\cdot\mathrm{lbf/ft}}{\mathrm{slug}}\right) = 1.12\ \mathrm{lbf} \longleftarrow W_{\mathrm{BG}}$$

In EE units, we use the form $W = mg/g_c$, and using the definition of g_c,

$$W_{\mathrm{EE}} = 1.12\ \mathrm{lbm} \times 32.2\ \frac{\mathrm{ft}}{\mathrm{s}^2} \times \frac{1}{g_c} = \frac{36.1}{g_c}\ \frac{\mathrm{lbm}\cdot\mathrm{ft}}{\mathrm{s}^2}$$

$$= 36.1\left(\frac{\mathrm{lbm}\cdot\mathrm{ft}}{\mathrm{s}^2}\right)\left(\frac{\mathrm{lbf}\cdot\mathrm{s}^2}{32.2\ \mathrm{ft}\cdot\mathrm{lbm}}\right) = 1.12\ \mathrm{lbf} \longleftarrow W_{\mathrm{EE}}$$

This problem illustrates:

- ✓ Conversions from SI to BG and EE systems.
- ✓ Use of g_c in the EE system.

Notes: The student may feel this example involves a lot of unnecessary calculation details (e.g., a factor of 32.2 appears, then disappears), but it cannot be stressed enough that such steps should always be explicitly written out to minimize errors—if you do not write all steps and units down, it is just too easy, for example, to multiply by a conversion factor when you should be dividing by it. For the weights in SI, BG, and EE units, we could alternatively have looked up the conversion from newton to lbf.

Dimensional Consistency and "Engineering" Equations

In engineering, we strive to make equations and formulas have consistent dimensions. That is, each term in an equation, and obviously both sides of the equation, should be reducible to the same dimensions. For example, a very important equation we will derive later on is the Bernoulli equation

$$\frac{p_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{p_2}{\rho} + \frac{V_2^2}{2} + gz_2$$

which relates the pressure p, velocity V, and elevation z between points 1 and 2 along a streamline for a steady, frictionless incompressible flow (density ρ). This equation is dimensionally consistent because each term in the equation can be reduced to dimensions of L^2/t^2 (the pressure term dimensions are FL/M, but from Newton's law we find $F = M/Lt^2$, so $FL/M = ML^2/Mt^2 = L^2/t^2$).

Almost all equations you are likely to encounter will be dimensionally consistent. However, you should be alert to some still commonly used equations that are not; these are often "engineering" equations derived many years ago, or are empirical (based on experiment rather than theory), or are proprietary equations used in a particular industry or company. For example, civil engineers often use the semi-empirical Manning equation

$$V = \frac{R_h^{2/3} S_0^{1/2}}{n}$$

which gives the flow speed V in an open channel (such as a canal) as a function of the hydraulic radius R_h (which is a measure of the flow cross-section and contact surface area), the channel slope S_0, and a constant n (the Manning resistance coefficient). The value of this constant depends on the surface condition of the channel. For example, for a canal made from unfinished concrete, most references give $n \approx 0.014$. Unfortunately, the equation is dimensionally inconsistent! For the right side of the equation, R_h has dimensions L, and S_0 is dimensionless, so with a dimensionless constant n, we end up with dimensions of $L^{2/3}$; for the left side of the equation the dimensions must be L/t! A user of the equation is supposed to know that the values of n provided in most references will give correct results *only* if we ignore the dimensional inconsistency, always use R_h in meters, and interpret V to be in m/s! (The alert student will realize that this means that even though handbooks provide n values as just constants, they must have units of $\mathrm{s/m^{1/3}}$.) Because the equation is dimensionally inconsistent, using the *same* value for n with R_h in ft does *not* give the correct value for V in ft/s.

A second type of problem is one in which the dimensions of an equation are consistent but use of units is not. The commonly used EER of an air conditioner is

$$EER = \frac{\text{cooling rate}}{\text{electrical input}}$$

which indicates how efficiently the AC works—a higher EER value indicates better performance. The equation *is* dimensionally consistent, with the EER being dimensionless (the cooling rate and electrical input are both measured in energy/time). However, it is *used*, in a sense, incorrectly, because the *units* traditionally used in it are not consistent. For example, a good EER value is 10, which would appear to imply you receive, say, 10 kW of cooling for each 1 kW of electrical power. In fact, an EER of 10 means you receive 10 *Btu/hr* of cooling for each 1 *W* of electrical power! Manufacturers, retailers, and customers all use the EER, in a sense, incorrectly in that they quote an EER of, say, 10, rather than the correct way, of 10 Btu/hr/W. (The EER, as used, is an everyday, inconsistent unit version of the coefficient of performance, COP, studied in thermodynamics.)

The two examples above illustrate the dangers in using certain equations. Almost all the equations encountered in this text will be dimensionally consistent, but you should be aware of the occasional troublesome equation you will encounter in your engineering studies.

As a final note on units, we stated earlier that we will use SI and BG units in this text. You will become very familiar with their use through using this text but should be aware that many of the units used, although they are scientifically and engineering-wise correct, are nevertheless not units you will use in everyday activities, and vice versa; we do not recommend asking your grocer to give you, say, 22 newtons, or 0.16 slugs, of potatoes; nor should you be expected to immediately know what, say, a motor oil viscosity of 5W20 means!

SI units and prefixes, other defined units, and useful conversions are given in Appendix G.

Analysis of Experimental Error 1.7

Most consumers are unaware of it but, as with most foodstuffs, soft drink containers are filled to plus or minus a certain amount, as allowed by law. Because it is difficult to precisely measure the filling of a container in a rapid production process, a 12-fl-oz container may actually contain 12.1, or 12.7, fl oz. The manufacturer is never supposed to supply less than the specified amount; and it will reduce profits if it is unnecessarily generous. Similarly, the supplier of components for the interior of a car must satisfy minimum and maximum dimensions (each component has what are called tolerances) so that the final appearance of the interior is visually appealing. Engineers performing experiments must measure not just data but also the uncertainties in their

measurements. They must also somehow determine how these uncertainties affect the uncertainty in the final result.

All of these examples illustrate the importance of *experimental uncertainty*, that is, the study of uncertainties in measurements and their effect on overall results. There is always a trade-off in experimental work or in manufacturing: We can reduce the uncertainties to a desired level, but the smaller the uncertainty (the more precise the measurement or experiment), the more expensive the procedure will be. Furthermore, in a complex manufacture or experiment, it is not always easy to see which measurement uncertainty has the biggest influence on the final outcome.

Anyone involved in manufacturing, or in experimental work, should understand experimental uncertainties. Appendix F has details on this topic; there is a selection of problems on this topic at the end of this chapter.

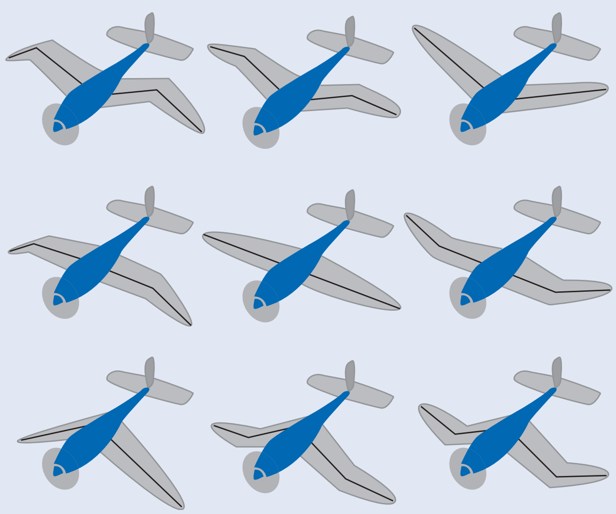
1.8 Summary

In this chapter we introduced or reviewed a number of basic concepts and definitions, including:

- ✓ How fluids are defined, and the no-slip condition
- ✓ System/control volume concepts
- ✓ Lagrangian and Eulerian descriptions
- ✓ Units and dimensions (including SI, British Gravitational, and English Engineering systems)
- ✓ Experimental uncertainty

Case Study

"Fly Like a Bird"



The airplane with various instantaneous wing shapes. (Courtesy of Dr. Rick Lind, University of Florida.)

At the end of each chapter, we present a case study: an interesting development in fluid mechanics chosen to illustrate that the field is constantly evolving.

No airplane, or airplane model, flies like a bird; aircraft all have fixed wings when in flight, whereas birds are (almost) constantly flapping away! One reason for this is that airplane and model wings must support relatively significant weight and are therefore thick and stiff; another reason is that we don't yet fully understand bird flight! Engineers at the University of Florida in Gainesville, led by researcher Rick Lind, have gone back to the drawing board and have developed a small surveillance aircraft (2-ft wingspan, weight a total of $1\frac{1}{2}$ lb) that can change its wing shape during flight. While it is not true bird flight (the main propulsion is through a propeller), it is a radical departure from current airplane design. The airplane can change, for example, from an M shape wing configuration (very stable for gliding) to a W shape (for high maneuverability). It is amazingly dexterous: It can turn three rolls in less than a second (comparable to an F-15 fighter!), and its flight is sufficiently birdlike that it has attracted sparrows (friendly) and crows (unfriendly). Possible uses are in military surveillance, detection of biological agents in congested urban areas, and environmental studies in difficult airspaces such as forests.

Problems

Definition of a Fluid: Basic Equations

1.1 A number of common substances are

Tar	Sand
"Silly Putty"	Jello
Modeling clay	Toothpaste
Wax	Shaving cream

Some of these materials exhibit characteristics of both solid and fluid behavior under different conditions. Explain and give examples.

1.2 Give a word statement of each of the five basic conservation laws stated in Section 1.4, as they apply to a system.

Methods of Analysis

1.3 The barrel of a bicycle tire pump becomes quite warm during use. Explain the mechanisms responsible for the temperature increase.

1.4 Discuss the physics of skipping a stone across the water surface of a lake. Compare these mechanisms with a stone as it bounces after being thrown along a roadway.

1.5 Make a guess at the order of magnitude of the mass (e.g., 0.01, 0.1, 1.0, 10, 100, or 1000 lbm or kg) of standard air that is in a room 10 ft by 10 ft by 8 ft, and then compute this mass in lbm and kg to see how close your estimate was.

1.6 A spherical tank of inside diameter 16 ft contains compressed oxygen at 1000 psia and 77°F. What is the mass of the oxygen?

1.7 Very small particles moving in fluids are known to experience a drag force proportional to speed. Consider a particle of net weight W dropped in a fluid. The particle experiences a drag force, $F_D = kV$, where V is the particle speed. Determine the time required for the particle to accelerate from rest to 95 percent of its terminal speed, V_t, in terms of k, W, and g.

1.8 Consider again the small particle of Problem 1.7. Express the distance required to reach 95 percent of its terminal speed in percent terms of g, k, and W.

1.9 A cylindrical tank must be designed to contain 5 kg of compressed nitrogen at a pressure of 200 atm (gage) and 20°C. The design constraints are that the length must be twice the diameter and the wall thickness must be 0.5 cm. What are the external dimensions?

1.10 In a combustion process, gasoline particles are to be dropped in air at 200°F. The particles must drop at least 10 in. in 1 s. Find the diameter d of droplets required for this. (The drag on these particles is given by $F_D = \pi\mu Vd$, where V is the particle speed and μ is the air viscosity. To solve this problem, use *Excel's Goal Seek*.)

1.11 For a small particle of styrofoam (1 lbm/ft^3) (spherical, with diameter $d = 0.3$ mm) falling in standard air at speed V, the drag is given by $F_D = 3\pi\mu Vd$, where μ is the air viscosity. Find the maximum speed starting from rest, and the time it takes to reach 95 percent of this speed. Plot the speed as a function of time.

1.12 In a pollution control experiment, minute solid particles (typical mass 1×10^{-13} slug) are dropped in air. The terminal speed of the particles is measured to be 0.2 ft/s. The drag of these particles is given by $F_D = kV$, where V is the instantaneous particle speed. Find the value of the constant k. Find the time required to reach 99 percent of terminal speed.

1.13 For Problem 1.12, find the distance the particles travel before reaching 99 percent of terminal speed. Plot the distance traveled as a function of time.

1.14 A sky diver with a mass of 70 kg jumps from an aircraft. The aerodynamic drag force acting on the sky diver is known to be $F_D = kV^2$, where $k = 0.25\ \text{N}\cdot\text{s}^2/\text{m}^2$. Determine the maximum speed of free fall for the sky diver and the speed reached after 100 m of fall. Plot the speed of the sky diver as a function of time and as a function of distance fallen.

1.15 For Problem 1.14, the initial horizontal speed of the sky diver is 70 m/s. As she falls, the k value for the vertical drag remains as before, but the value for horizontal motion is $k = 0.05\ \text{N}\cdot\text{s/m}^2$. Compute and plot the 2D trajectory of the sky diver.

1.16 The English perfected the longbow as a weapon after the Medieval period. In the hands of a skilled archer, the longbow was reputed to be accurate at ranges to 100 m or more. If the maximum altitude of an arrow is less than $h = 10$ m while traveling to a target 100 m away from the archer, and neglecting air resistance, estimate the speed and angle at which the arrow must leave the bow. Plot the required release speed and angle as a function of height h.

Dimensions and Units

1.17 For each quantity listed, indicate dimensions using mass as a primary dimension, and give typical SI and English units:
(a) Power
(b) Pressure
(c) Modulus of elasticity
(d) Angular velocity
(e) Energy
(f) Moment of a force
(g) Momentum
(h) Shear stress
(i) Strain
(j) Angular momentum

1.18 For each quantity listed, indicate dimensions using force as a primary dimension, and give typical SI and English units:
(a) Power
(b) Pressure
(c) Modulus of elasticity
(d) Angular velocity
(e) Energy
(f) Momentum
(g) Shear stress
(h) Specific heat

(i) Thermal expansion coefficient
(j) Angular momentum

1.19 Derive the following conversion factors:
(a) Convert a viscosity of 1 m^2/s to ft^2/s.
(b) Convert a power of 100 W to horsepower.
(c) Convert a specific energy of 1 kJ/kg to Btu/lbm.

1.20 Derive the following conversion factors:
(a) Convert a pressure of 1 psi to kPa.
(b) Convert a volume of 1 liter to gallons.
(c) Convert a viscosity of 1 $lbf \cdot s/ft^2$ to $N \cdot s/m^2$.

1.21 Derive the following conversion factors:
(a) Convert a specific heat of 4.18 $kJ/kg \cdot K$ to $Btu/lbm \cdot {}^\circ R$.
(b) Convert a speed of 30 m/s to mph.
(c) Convert a volume of 5.0 L to in^3.

1.22 Express the following in SI units:
(a) 5 $acre \cdot ft$
(b) 150 in^3/s
(c) 3 gpm
(d) 3 mph/s

1.23 Express the following in SI units:
(a) 100 cfm (ft^3/min)
(b) 5 gal
(c) 65 mph
(d) 5.4 acres

1.24 Express the following in BG units:
(a) 50 m^2
(b) 250 cc
(c) 100 kW
(d) 5 kg/m^2

1.25 Express the following in BG units:
(a) 180 cc/min
(b) 300 $kW \cdot hr$
(c) 50 $N \cdot s/m^2$
(d) 40 $m^2 \cdot hr$

1.26 While you're waiting for the ribs to cook, you muse about the propane tank of your barbecue. You're curious about the volume of propane versus the actual tank size. Find the liquid propane volume when full (the weight of the propane is specified on the tank). Compare this to the tank volume (take some measurements, and approximate the tank shape as a cylinder with a hemisphere on each end). Explain the discrepancy.

1.27 A farmer needs 4 cm of rain per week on his farm, with 10 hectares of crops. If there is a drought, how much water (L/min) will have to be supplied to maintain his crops?

1.28 Derive the following conversion factors:
(a) Convert a volume flow rate in cubic inches per minute to cubic millimeters per minute.
(b) Convert a volume flow rate in cubic meters per second to gallons per minute (gpm).
(c) Convert a volume flow rate in liters per minute to gpm.
(d) Convert a volume flow rate of air in standard cubic feet per minute (SCFM) to cubic meters per hour. A standard cubic foot of gas occupies one cubic foot at standard temperature and pressure ($T = 15^\circ C$ and $p = 101.3$ kPa absolute).

1.29 The density of mercury is given as 26.3 $slug/ft^3$. Calculate the specific gravity and the specific volume in m^3/kg of the mercury. Calculate the specific weight in lbf/ft^3 on Earth and on the moon. Acceleration of gravity on the moon is 5.47 ft/s^2.

1.30 The kilogram force is commonly used in Europe as a unit of force. (As in the U.S. customary system, where 1 lbf is the force exerted by a mass of 1 lbm in standard gravity, 1 kgf is the force exerted by a mass of 1 kg in standard gravity.) Moderate pressures, such as those for auto or truck tires, are conveniently expressed in units of kgf/cm^2. Convert 32 psig to these units.

1.31 In Section 1.6 we learned that the Manning equation computes the flow speed V (m/s) in a canal made from unfinished concrete, given the hydraulic radius R_h (m), the channel slope S_0, and a Manning resistance coefficient constant value $n \approx 0.014$. For a canal with $R_h = 7.5$ m and a slope of 1/10, find the flow speed. Compare this result with that obtained using the same n value, but with R_h first converted to ft, with the answer assumed to be in ft/s. Finally, find the value of n if we wish to *correctly* use the equation for BG units (and compute V to check!).

1.32 From thermodynamics, we know that the coefficient of performance of an ideal air conditioner (COP_{ideal}) is given by

$$COP_{\text{ideal}} = \frac{T_L}{T_H - T_L}$$

where T_L and T_H are the room and outside temperatures (absolute). If an AC is to keep a room at 20°C when it is 40°C outside, find the COP_{ideal}. Convert to an EER value, and compare this to a typical Energy Star–compliant EER value.

1.33 The maximum theoretical flow rate (slug/s) through a supersonic nozzle is

$$\dot{m}_{\max} = 2.38 \frac{A_t p_0}{\sqrt{T_0}}$$

where A_t (ft^2) is the nozzle throat area, p_0 (psi) is the tank pressure, and T_0 (°R) is the tank temperature. Is this equation dimensionally correct? If not, find the units of the 2.38 term. Write the equivalent equation in SI units.

1.34 The mean free path λ of a molecule of gas is the average distance it travels before collision with another molecule. It is given by

$$\lambda = C \frac{m}{\rho d^2}$$

where m and d are the molecule's mass and diameter, respectively, and ρ is the gas density. What are the dimensions of constant C for a dimensionally consistent equation?

1.35 In Chapter 9 we will study aerodynamics and learn that the drag force F_D on a body is given by

$$F_D = \frac{1}{2} \rho V^2 A C_D$$

Hence the drag depends on speed V, fluid density ρ, and body size (indicated by frontal area A) and shape (indicated by drag coefficient C_D). What are the dimensions of C_D?

1.36 A container weighs 3.5 lbf when empty. When filled with water at 90°F, the mass of the container and its contents is 2.5 slug. Find the weight of water in the container, and its volume in cubic feet, using data from Appendix A.

1.37 An important equation in the theory of vibrations is

$$m\frac{d^2x}{dt^2} + c\frac{dx}{dt} + kx = f(t)$$

where m (kg) is the mass and x (m) is the position at time t (s). For a dimensionally consistent equation, what are the dimensions of c, k, and f? What would be suitable units for c, k, and f in the SI and BG systems?

1.38 A parameter that is often used in describing pump performance is the specific speed, $N_{S_{cu}}$, given by

$$N_{S_{cu}} = \frac{N(\text{rpm})[Q(\text{gpm})]^{1/2}}{[H(\text{ft})]^{3/4}}$$

What are the units of specific speed? A particular pump has a specific speed of 2000. What will be the specific speed in SI units (angular velocity in rad/s)?

1.39 A particular pump has an "engineering" equation form of the performance characteristic equation given by H (ft) $= 1.5 - 4.5 \times 10^{-5}$ $[Q$ (gpm)$]^2$, relating the head H and flow rate Q. What are the units of the coefficients 1.5 and 4.5×10^{-5}? Derive an SI version of this equation.

Analysis of Experimental Error

1.40 Calculate the density of standard air in a laboratory from the ideal gas equation of state. Estimate the experimental uncertainty in the air density calculated for standard conditions (29.9 in. of mercury and 59°F) if the uncertainty in measuring the barometer height is ±0.1 in. of mercury and the uncertainty in measuring temperature is ±0.5°F. (Note that 29.9 in. of mercury corresponds to 14.7 psia.)

1.41 Repeat the calculation of uncertainty described in Problem 1.40 for air in a hot air balloon. Assume the measured barometer height is 759 mm of mercury with an uncertainty of ±1 mm of mercury and the temperature is 60°C with an uncertainty of ±1°C. [Note that 759 mm of mercury corresponds to 101 kPa (abs).]

1.42 The mass of the standard American golf ball is 1.62 ± 0.01 oz and its mean diameter is 1.68 ± 0.01 in. Determine the density and specific gravity of the American golf ball. Estimate the uncertainties in the calculated values.

1.43 A can of pet food has the following internal dimensions: 102 mm height and 73 mm diameter (each ±1 mm at odds of 20 to 1). The label lists the mass of the contents as 397 g. Evaluate the magnitude and estimated uncertainty of the density of the pet food if the mass value is accurate to ±1 g at the same odds.

1.44 The mass flow rate in a water flow system determined by collecting the discharge over a timed interval is 0.2 kg/s. The scales used can be read to the nearest 0.05 kg and the stopwatch is accurate to 0.2 s. Estimate the precision with which the flow rate can be calculated for time intervals of (a) 10 s and (b) 1 min.

1.45 The mass flow rate of water in a tube is measured using a beaker to catch water during a timed interval. The nominal mass flow rate is 100 g/s. Assume that mass is measured using a balance with a least count of 1 g and a maximum capacity of 1 kg, and that the timer has a least count of 0.1 s. Estimate the time intervals and uncertainties in measured mass flow rate that would result from using 100, 500, and 1000 mL beakers. Would there be any advantage in using the largest beaker? Assume the tare mass of the empty 1000 mL beaker is 500 g.

1.46 The mass of the standard British golf ball is 45.9 ± 0.3 g and its mean diameter is 41.1 ± 0.3 mm. Determine the density and specific gravity of the British golf ball. Estimate the uncertainties in the calculated values.

1.47 The estimated dimensions of a soda can are $D = 66.0 \pm 0.5$ mm and $H = 110 \pm 0.5$ mm. Measure the mass of a full can and an empty can using a kitchen scale or postal scale. Estimate the volume of soda contained in the can. From your measurements estimate the depth to which the can is filled and the uncertainty in the estimate. Assume the value of SG = 1.055, as supplied by the bottler.

1.48 From Appendix A, the viscosity μ (N · s/m^2) of water at temperature T (K) can be computed from $\mu = A10^{B/(T-C)}$, where $A = 2.414 \times 10^{-5}$ N · s/m^2, $B = 247.8$ K, and $C = 140$ K. Determine the viscosity of water at 30°C, and estimate its uncertainty if the uncertainty in temperature measurement is ±0.5°C.

1.49 Using the nominal dimensions of the soda can given in Problem 1.47, determine the precision with which the diameter and height must be measured to estimate the volume of the can within an uncertainty of ±0.5 percent.

1.50 An enthusiast magazine publishes data from its road tests on the lateral acceleration capability of cars. The measurements are made using a 150-ft-diameter skid pad. Assume the vehicle path deviates from the circle by ±2 ft and that the vehicle speed is read from a fifth-wheel speed-measuring system to ±0.5 mph. Estimate the experimental uncertainty in a reported lateral acceleration of 0.7 g. How would you improve the experimental procedure to reduce the uncertainty?

1.51 The height of a building may be estimated by measuring the horizontal distance to a point on the ground and the angle from this point to the top of the building. Assuming these measurements are $L = 100 \pm 0.5$ ft and $\theta = 30 \pm 0.2°$, estimate the height H of the building and the uncertainty in the estimate. For the same building height and measurement uncertainties, use *Excel's Solver* to determine the angle (and the corresponding distance from the building) at which measurements should be made to minimize the uncertainty in estimated height. Evaluate and plot the optimum measurement angle as a function of building height for $50 \le H \le 1000$ ft.

1.52 An American golf ball is described in Problem 1.42 Assuming the measured mass and its uncertainty as given, determine the precision to which the diameter of the ball must be measured so the density of the ball may be estimated within an uncertainty of ±1 percent.

1.53 A syringe pump is to dispense liquid at a flow rate of 100 mL/min. The design for the piston drive is such that the uncertainty of the piston speed is 0.001 in./min, and the cylinder bore diameter has a maximum uncertainty of 0.0005 in. Plot the uncertainty in the flow rate as a function of cylinder bore. Find the combination of piston speed and bore that minimizes the uncertainty in the flow rate.

2

Fundamental Concepts

Case Study in Energy and the Environment

Ocean Power

We're not used to thinking of them this way, but the oceans are a huge repository of solar energy (and energy due to the moon's motion). The solar energy storage is initially thermal in nature, as the water surface is heated during the day. When the water cools overnight, thermal gradients are created that ultimately lead to ocean currents (as well as winds) containing huge amounts of energy. According to a 2009 U.S. Department of Energy study titled "Ocean Energy Technology," there are four types of ocean energy conversion: *wave energy, tidal energy, marine current energy*, and *ocean thermal energy conversion*.

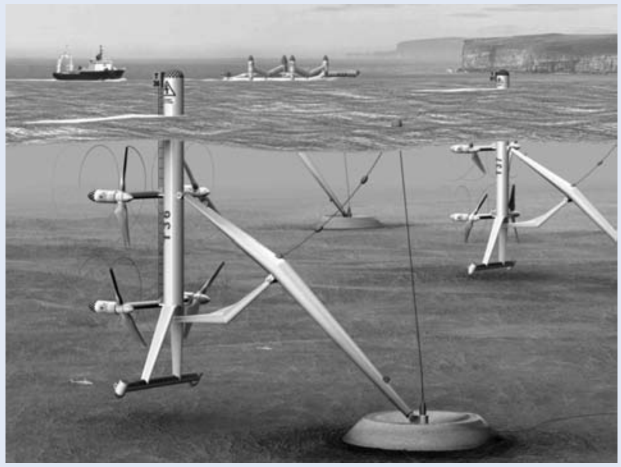
The total power from waves believed to be available is about 2.7 TW, of which it is currently practical to extract 500 GW (500×10^9 W). Bear in mind that we mentioned in Chapter 1 that *total* power consumption by humans was about 16 TW (as of 2006), so at best wave power could supply about 3 percent of human needs using current technology. These devices work by either floating on the surface of the ocean or by being moored to the ocean floor. Many of these devices rely on buoyancy forces, which we will discuss in Chapter 3. For example, a device that floats on the surface may have joints hinged together that bend with the waves; this bending motion pumps fluid through turbines and creates electric power. Alternatively, stationary tethered devices use pressure fluctuations produced in long tubes from the waves swelling up and down; the bobbing motion drives a turbine. Wave energy is already

reaching fairly advanced levels of development, with a number of companies being involved.

Tidal energy uses the 12-hr cycle due to the gravitational force of the moon; the difference in water height from low to high tide is an extractable form of potential energy. For example, water can be captured by using a barrier across an estuary during high tide and by forcing the water through a turbine during low tide. Alternatively, as shown in the figure, turbine systems could be mounted in such a way that they swing with the tide, extracting energy when the tide comes in and goes out. There are only about 20 locations on earth with tides sufficiently high to make tidal energy practical. The Bay of Fundy between Maine and Nova Scotia features the highest tides in the world, reaching 17 m (56 ft). This area alone could produce up to 15 GW of power. The total wave energy power believed to be available is about 2.5 TW, of which, with current technology, it is practical to extract only about 65 GW.

Marine current energy is that due to ocean currents (which in turn are generated by solar heating and by the winds—ultimately solar in origin—as well as by the Earth's rotation). About 5 TW of power is believed to be available, of which it is currently practical to extract 450 GW; at best, this energy source will supply something less than 5 percent of total current needs. In the United States, it is most abundant off the coast of Florida in the flow known as the Gulf Stream. Kinetic energy can be captured from the Gulf Stream and other currents with submerged turbines that are very similar in appearance to miniature wind turbines. As with wind turbines, the continuous movement of the marine current moves the rotor blades to generate electric power. Turbines will be discussed in some detail in Chapter 10.

Ocean thermal energy conversion (OTEC), uses the ocean temperature difference between surface water and that at depths lower than 1000 m to extract energy. The temperature of ocean water at a depth of 1000 m is just above freezing; a temperature difference of as little as 20°C (36°F) can yield usable energy. (You can figure out the minimum surface temperature required!) The warm surface water can be used as a heat source to evaporate a fluid such as ammonia, which can drive a turbine, and the deep water acts as a heat sink. Because of the temperatures involved, such devices will have a very low theoretical efficiency, but the amount of stored thermal energy is huge—about 200 TW of power!



Proposed tidal turbines.

Yet another form of ocean energy (ultimately traceable to solar energy) is that due to the variability of salinity due to water evaporation. When salty ocean water (brine) is separated from fresh water by a semipermeable membrane, a pressure gradient builds up across the membrane (osmotic pressure). We will learn in this text that a pressure gradient can be used as a driving force for energy generation. The exploitation of this energy is called *salinity gradient energy conversion*. This is a future technology with huge potential. There is about 1000 TW of energy available, or about 60 times total worldwide power usage!

We shall discuss some interesting developments in several of these energy conversion methods in *Case Studies in Energy and the Environment* in subsequent chapters.

In Chapter 1 we discussed in general terms what fluid mechanics is about, and described some of the approaches we will use in analyzing fluid mechanics problems. In this chapter we will be more specific in defining some important properties of fluids and ways in which flows can be described and characterized.

Fluid as a Continuum 2.1

We are all familiar with fluids—the most common being air and water—and we experience them as being "smooth," i.e., as being a continuous medium. Unless we use

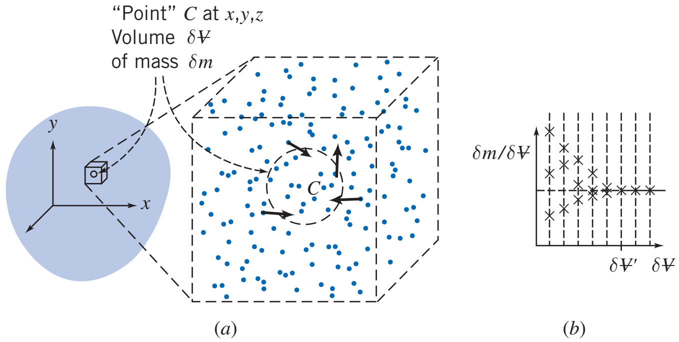


Fig. 2.1 Definition of density at a point.

VIDEO

Fluid as a Continuum.

specialized equipment, we are not aware of the underlying molecular nature of fluids. This molecular structure is one in which the mass is *not* continuously distributed in space, but is concentrated in molecules that are separated by relatively large regions of empty space. The sketch in Fig. 2.1*a* shows a schematic representation of this. A region of space "filled" by a stationary fluid (e.g., air, treated as a single gas) looks like a continuous medium, but if we zoom in on a very small cube of it, we can see that we mostly have empty space, with gas molecules scattered around, moving at high speed (indicated by the gas temperature). Note that the size of the gas molecules is greatly exaggerated (they would be almost invisible even at this scale) and that we have placed velocity vectors only on a small sample. We wish to ask: What is the minimum volume, $\delta\mathcal{V}'$, that a "point" C must be, so that we can talk about continuous fluid properties such as the density at a point? In other words, under what circumstances can a fluid be treated as a *continuum*, for which, by definition, properties vary smoothly from point to point? This is an important question because the concept of a continuum is the basis of classical fluid mechanics.

Consider how we determine the density at a point. Density is defined as mass per unit volume; in Fig. 2.1*a* the mass δm will be given by the instantaneous number of molecules in $\delta\mathcal{V}$ (and the mass of each molecule), so the average density in volume $\delta\mathcal{V}$ is given by $\rho = \delta m/\delta\mathcal{V}$. We say "average" because the number of molecules in $\delta\mathcal{V}$, and hence the density, fluctuates. For example, if the gas in Fig. 2.1*a* was air at standard temperature and pressure (STP[1]) and the volume $\delta\mathcal{V}$ was a sphere of diameter 0.01 μm, there might be 15 molecules in $\delta\mathcal{V}$ (as shown), but an instant later there might be 17 (three might enter while one leaves). Hence the density at "point" C randomly fluctuates in time, as shown in Fig. 2.1*b*. In this figure, each vertical dashed line represents a specific chosen volume, $\delta\mathcal{V}$, and each data point represents the measured density at an instant. For very small volumes, the density varies greatly, but above a certain volume, $\delta\mathcal{V}'$, the density becomes stable—the volume now encloses a huge number of molecules. For example, if $\delta\mathcal{V} = 0.001\ \text{mm}^3$ (about the size of a grain of sand), there will on average be 2.5×10^{13} molecules present. Hence we can conclude that air at STP (and other gases, and liquids) can be treated as a continuous medium as long as we consider a "point" to be no smaller than about this size; this is sufficiently precise for most engineering applications.

The concept of a continuum is the basis of classical fluid mechanics. The continuum assumption is valid in treating the behavior of fluids under normal conditions. It only breaks down when the mean free path of the molecules[2] becomes the same order of magnitude as the smallest significant characteristic dimension of the problem.

[1]STP for air are 15°C (59°F) and 101.3 kPa absolute (14.696 psia), respectively.

[2]Approximately 6×10^{-8} m at STP (Standard Temperature and Pressure) for gas molecules that show ideal gas behavior [1].

This occurs in such specialized problems as rarefied gas flow (e.g., as encountered in flights into the upper reaches of the atmosphere). For these specialized cases (not covered in this text) we must abandon the concept of a continuum in favor of the microscopic and statistical points of view.

As a consequence of the continuum assumption, each fluid property is assumed to have a definite value at every point in space. Thus fluid properties such as density, temperature, velocity, and so on are considered to be continuous functions of position and time. For example, we now have a workable definition of density at a point,

$$\rho \equiv \lim_{\delta \mathcal{V} \to \delta \mathcal{V}'} \frac{\delta m}{\delta \mathcal{V}} \tag{2.1}$$

Since point C was arbitrary, the density at any other point in the fluid could be determined in the same manner. If density was measured simultaneously at an infinite number of points in the fluid, we would obtain an expression for the density distribution as a function of the space coordinates, $\rho = \rho(x, y, z)$, at the given instant.

The density at a point may also vary with time (as a result of work done on or by the fluid and/or heat transfer to the fluid). Thus the complete representation of density (the *field* representation) is given by

$$\rho = \rho(x, y, z, t) \tag{2.2}$$

Since density is a scalar quantity, requiring only the specification of a magnitude for a complete description, the field represented by Eq. 2.2 is a scalar field.

An alternative way of expressing the density of a substance (solid or fluid) is to compare it to an accepted reference value, typically the maximum density of water, ρ_{H_2O} (1000 kg/m^3 at 4°C or 1.94 slug/ft^3 at 39°F). Thus, the *specific gravity*, SG, of a substance is expressed as

$$SG = \frac{\rho}{\rho_{H_2O}} \tag{2.3}$$

For example, the SG of mercury is typically 13.6—mercury is 13.6 times as dense as water. Appendix A contains specific gravity data for selected engineering materials. The specific gravity of liquids is a function of temperature; for most liquids specific gravity decreases with increasing temperature.

The *specific weight*, γ, of a substance is another useful material property. It is defined as the weight of a substance per unit volume and given as

$$\gamma = \frac{mg}{\mathcal{V}} \rightarrow \gamma = \rho g \tag{2.4}$$

For example, the specific weight of water is approximately 9.81 kN/m^3 (62.4 lbf/ft^3).

Velocity Field 2.2

In the previous section we saw that the continuum assumption led directly to the notion of the density field. Other fluid properties also may be described by fields.

A very important property defined by a field is the velocity field, given by

$$\vec{V} = \vec{V}(x, y, z, t) \tag{2.5}$$

Velocity is a vector quantity, requiring a magnitude and direction for a complete description, so the velocity field (Eq. 2.5) is a vector field.

The velocity vector, $\vec{V}$, also can be written in terms of its three scalar components. Denoting the components in the x, y, and z directions by u, v, and w, then

$$\vec{V} = u\hat{i} + v\hat{j} + w\hat{k} \tag{2.6}$$

In general, each component, u, v, and w, will be a function of x, y, z, and t.

We need to be clear on what $\vec{V}(x, y, z, t)$ measures: It indicates the velocity of a fluid particle that is passing through the point x, y, z at time instant t, in the Eulerian sense. We can keep measuring the velocity at the same point or choose any other point x, y, z at the next time instant; the point x, y, z is *not* the ongoing position of an *individual* particle, but a point we choose to look at. (Hence x, y, and z are independent variables. In Chapter 5 we will discuss the *material derivative* of velocity, in which we *choose* $x = x_p(t), y = y_p(t)$, and $z = z_p(t)$, where $x_p(t)$, $y_p(t)$, $z_p(t)$ *is* the position of a specific particle.) We conclude that $\vec{V}(x, y, z, t)$ should be thought of as the velocity field of all particles, not just the velocity of an individual particle.

If properties at every point in a flow field do not change with time, the flow is termed *steady*. Stated mathematically, the definition of steady flow is

$$\frac{\partial \eta}{\partial t} = 0$$

where η represents any fluid property. Hence, for steady flow,

$$\frac{\partial \rho}{\partial t} = 0 \quad \text{or} \quad \rho = \rho(x, y, z)$$

and

$$\frac{\partial \vec{V}}{\partial t} = 0 \quad \text{or} \quad \vec{V} = \vec{V}(x, y, z)$$

In steady flow, any property may vary from point to point in the field, but all properties remain constant with time at every point.

One-, Two-, and Three-Dimensional Flows

A flow is classified as one-, two-, or three-dimensional depending on the number of space coordinates required to specify the velocity field.[3] Equation 2.5 indicates that the velocity field may be a function of three space coordinates and time. Such a flow field is termed *three-dimensional* (it is also *unsteady*) because the velocity at any point in the flow field depends on the three coordinates required to locate the point in space.

Although most flow fields are inherently three-dimensional, analysis based on fewer dimensions is frequently meaningful. Consider, for example, the steady flow through a long straight pipe that has a divergent section, as shown in Fig. 2.2. In this example, we are using cylindrical coordinates (r, θ, x). We will learn (in Chapter 8) that under certain circumstances (e.g., far from the entrance of the pipe and from the divergent section, where the flow can be quite complicated), the velocity distribution may be described by

$$u = u_{\max}\left[1 - \left(\frac{r}{R}\right)^2\right] \tag{2.7}$$

This is shown on the left of Fig. 2.2. The velocity $u(r)$ is a function of only one coordinate, and so the flow is one-dimensional. On the other hand, in the diverging

[3]Some authors choose to classify a flow as one-, two-, or three-dimensional on the basis of the number of space coordinates required to specify *all* fluid properties. In this text, classification of flow fields will be based on the number of space coordinates required to specify the velocity field only.

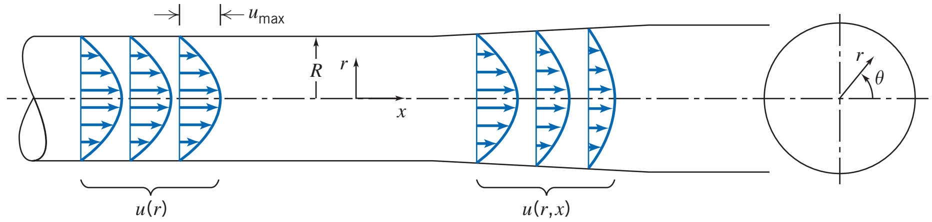


Fig. 2.2 Examples of one- and two-dimensional flows.

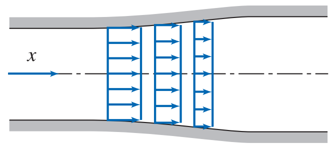


Fig. 2.3 Example of uniform flow at a section.

section, the velocity decreases in the x direction, and the flow becomes two-dimensional: $u = u(r, x)$.

As you might suspect, the complexity of analysis increases considerably with the number of dimensions of the flow field. For many problems encountered in engineering, a one-dimensional analysis is adequate to provide approximate solutions of engineering accuracy.

Since all fluids satisfying the continuum assumption must have zero relative velocity at a solid surface (to satisfy the no-slip condition), most flows are inherently two- or three-dimensional. To simplify the analysis it is often convenient to use the notion of *uniform flow* at a given cross section. In a flow that is uniform at a given cross section, the velocity is constant across any section normal to the flow. Under this assumption,[4] the two-dimensional flow of Fig. 2.2 is modeled as the flow shown in Fig. 2.3. In the flow of Fig. 2.3, the velocity field is a function of x alone, and thus the flow model is one-dimensional. (Other properties, such as density or pressure, also may be assumed uniform at a section, if appropriate.)

The term *uniform flow field* (as opposed to uniform flow at a cross section) is used to describe a flow in which the velocity is constant, i.e., independent of all space coordinates, throughout the entire flow field.

CLASSIC VIDEO

Flow Visualization.

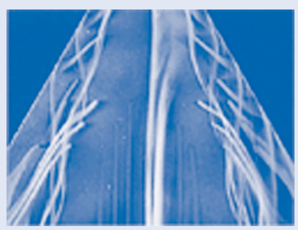
Timelines, Pathlines, Streaklines, and Streamlines

Airplane and auto companies and college engineering laboratories, among others, frequently use wind tunnels to visualize flow fields [2]. For example, Fig. 2.4 shows a flow pattern for flow around a car mounted in a wind tunnel, generated by releasing smoke into the flow at five fixed upstream points. Flow patterns can be visualized using timelines, pathlines, streaklines, or streamlines.

If a number of adjacent fluid particles in a flow field are marked at a given instant, they form a line in the fluid at that instant; this line is called a *timeline*. Subsequent observations of the line may provide information about the flow field. For example, in discussing the behavior of a fluid under the action of a constant shear force (Section 1.2) timelines were introduced to demonstrate the deformation of a fluid at successive instants.

VIDEO

Streaklines.



A *pathline* is the path or trajectory traced out by a moving fluid particle. To make a pathline visible, we might identify a fluid particle at a given instant, e.g., by the use of dye or smoke, and then take a long exposure photograph of its subsequent motion. The line traced out by the particle is a pathline. This approach might be used to study, for example, the trajectory of a contaminant leaving a smokestack.

On the other hand, we might choose to focus our attention on a fixed location in space and identify, again by the use of dye or smoke, all fluid particles passing through this point. After a short period of time we would have a number of identifiable fluid

[4]This may seem like an unrealistic simplification, but actually in many cases leads to useful results. Sweeping assumptions such as uniform flow at a cross section should always be reviewed carefully to be sure they provide a reasonable analytical model of the real flow.

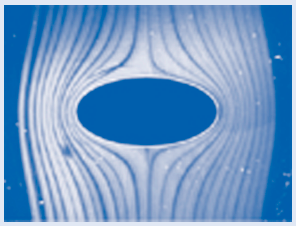
Fig. 2.4 Streaklines over an automobile in a wind tunnel. (Courtesy Audi AG.)

particles in the flow, all of which had, at some time, passed through one fixed location in space. The line joining these fluid particles is defined as a *streakline*.

Streamlines are lines drawn in the flow field so that at a given instant they are tangent to the direction of flow at every point in the flow field. Since the streamlines are tangent to the velocity vector at every point in the flow field, there can be no flow across a streamline. Streamlines are the most commonly used visualization technique. For example, they are used to study flow over an automobile in a computer simulation. The procedure used to obtain the equation for a streamline in two-dimensional flow is illustrated in Example 2.1.

VIDEO

Streamlines.



In steady flow, the velocity at each point in the flow field remains constant with time and, consequently, the streamline shapes do not vary from one instant to the next. This implies that a particle located on a given streamline will always move along the same streamline. Furthermore, consecutive particles passing through a fixed point in space will be on the same streamline and, subsequently, will remain on this streamline. Thus in a steady flow, pathlines, streaklines, and streamlines are identical lines in the flow field.

Figure 2.4 shows a photograph of five *streaklines* for flow over an automobile in a wind tunnel. A streakline is the line produced in a flow when all particles moving through a fixed point are marked in some way (e.g., using smoke). We can also define *streamlines*. These are lines drawn in the flow field so that *at a given instant* they are tangent to the direction of flow at every point in the flow field. Since the streamlines are tangent to the velocity vector at every point in the flow field, there is no flow across a streamline. *Pathlines* are as the name implies: They show, over time, the paths individual particles take (if you've seen time-lapse photos of nighttime traffic, you get the idea). Finally, *timelines* are created by marking a line in a flow and watching how it evolves over time.

We mentioned that Fig. 2.4 shows streaklines, but in fact the pattern shown also represents streamlines and pathlines! The steady pattern shown will exist as long as smoke is released from the five fixed points. If we were somehow to measure the velocity at all points at an instant, to generate streamlines, we'd get the same pattern; if we were instead to release only one smoke particle at each location, and video its motion over time, we'd see the particles follow the same curves. We conclude that for *steady* flow, streaklines, streamlines, and pathlines are identical.

Things are quite different for *unsteady* flow. For unsteady flow, streaklines, streamlines, and pathlines will in general have differing shapes. For example, consider holding a garden hose and swinging it side to side as water exits at high speed, as shown in Fig. 2.5. We obtain a continuous sheet of water. If we consider individual water particles, we see that each particle, once ejected, follows a straight-line path (here, for simplicity, we ignore gravity): The pathlines are straight lines, as shown. On the other hand, if we start injecting dye into the water as it exits the hose, we will generate a streakline, and this takes the shape of an expanding sine wave, as shown. Clearly, pathlines and streaklines do not coincide for this unsteady flow (we leave determination of streamlines to an exercise).

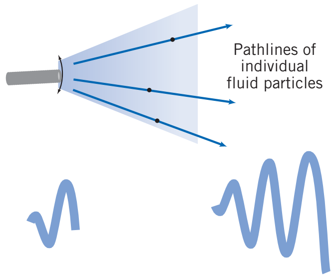


Fig. 2.5 Pathlines and streaklines for flow from the exit of an oscillating garden hose.

We can use the velocity field to derive the shapes of streaklines, pathlines, and streamlines. Starting with streamlines: Because the streamlines are parallel to the velocity vector, we can write (for 2D)

$$\left.\frac{dy}{dx}\right)_{\text{streamline}} = \frac{v(x, y)}{u(x, y)} \tag{2.8}$$

Note that streamlines are obtained at an instant in time; if the flow is unsteady, time t is held constant in Eq. 2.8. Solution of this equation gives the equation $y = y(x)$, with an undetermined integration constant, the value of which determines the particular streamline.

For pathlines (again considering 2D), we let $x = x_p(t)$ and $y = y_p(t)$, where $x_p(t)$ and $y_p(t)$ are the instantaneous coordinates of a specific particle. We then get

$$\left.\frac{dx}{dt}\right)_{\text{particle}} = u(x, y, t) \qquad \left.\frac{dy}{dt}\right)_{\text{particle}} = v(x, y, t) \tag{2.9}$$

The simultaneous solution of these equations gives the path of a particle in parametric form $x_p(t)$, $y_p(t)$.

The computation of streaklines is somewhat tricky. The first step is to compute the pathline of a particle (using Eqs. 2.9) that was released from the streak source point (coordinates x_0, y_0) at time t_0, in the form

$$x_{\text{particle}}(t) = x(t, x_0, y_0, t_0) \qquad y_{\text{particle}}(t) = y(t, x_0, y_0, t_0)$$

Then, instead of interpreting this as the position of a particle over time, we rewrite these equations as

$$x_{\text{streakline}}(t_0) = x(t, x_0, y_0, t_0) \qquad y_{\text{streakline}}(t_0) = y(t, x_0, y_0, t_0) \tag{2.10}$$

Equations 2.10 give the line generated (by time t) from a streak source at point (x_0, y_0). In these equations, t_0 (the release times of particles) is varied from 0 to t to show the instantaneous positions of all particles released up to time t!

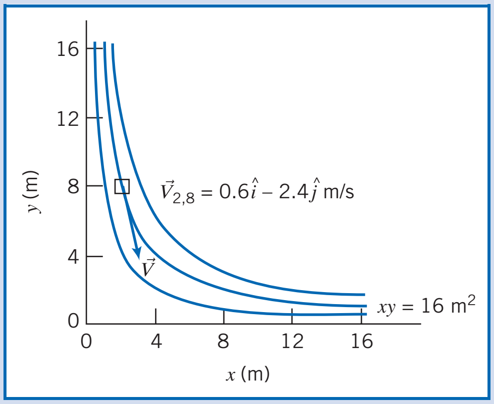
Example 2.1 STREAMLINES AND PATHLINES IN TWO-DIMENSIONAL FLOW

A velocity field is given by $\vec{V} = Ax\hat{i} - Ay\hat{j}$; the units of velocity are m/s; x and y are given in meters; $A = 0.3\ \text{s}^{-1}$.

(a) Obtain an equation for the streamlines in the xy plane.
(b) Plot the streamline passing through the point $(x_0, y_0) = (2, 8)$.
(c) Determine the velocity of a particle at the point (2, 8).
(d) If the particle passing through the point (x_0, y_0) is marked at time $t = 0$, determine the location of the particle at time $t = 6$ s.
(e) What is the velocity of this particle at time $t = 6$ s?
(f) Show that the equation of the particle path (the pathline) is the same as the equation of the streamline.

Given: Velocity field, $\vec{V} = Ax\hat{i} - Ay\hat{j}$; x and y in meters; $A = 0.3\ \text{s}^{-1}$.

Find: (a) Equation of the streamlines in the xy plane.
(b) Streamline plot through point (2, 8).
(c) Velocity of particle at point (2, 8).
(d) Position at $t = 6$ s of particle located at (2, 8) at $t = 0$.
(e) Velocity of particle at position found in (d).
(f) Equation of pathline of particle located at (2, 8) at $t = 0$.



Solution:

(a) Streamlines are lines drawn in the flow field such that, at a given instant, they are tangent to the direction of flow at every point. Consequently,

$$\left.\frac{dy}{dx}\right)_{\text{streamline}} = \frac{v}{u} = \frac{-Ay}{Ax} = \frac{-y}{x}$$

Separating variables and integrating, we obtain

$$\int \frac{dy}{y} = -\int \frac{dx}{x}$$

or

$$\ln y = -\ln x + c_1$$

This can be written as $xy = c$ ←

(b) For the streamline passing through the point $(x_0, y_0) = (2, 8)$ the constant, c, has a value of 16 and the equation of the streamline through the point (2, 8) is

$$xy = x_0 y_0 = 16\,\text{m}^2 \leftarrow$$

The plot is as sketched above.

(c) The velocity field is $\vec{V} = Ax\hat{i} - Ay\hat{j}$. At the point (2, 8) the velocity is

$$\vec{V} = A(x\hat{i} - y\hat{j}) = 0.3\text{s}^{-1}(2\hat{i} - 8\hat{j})\text{m} = 0.6\hat{i} - 2.4\hat{j}\,\text{m/s} \leftarrow$$

(d) A particle moving in the flow field will have velocity given by

$$\vec{V} = Ax\hat{i} - Ay\hat{j}$$

Thus

$$u_p = \frac{dx}{dt} = Ax \quad \text{and} \quad v_p = \frac{dy}{dt} = -Ay$$

Separating variables and integrating (in each equation) gives

$$\int_{x_0}^{x} \frac{dx}{x} = \int_0^t A\,dt \quad \text{and} \quad \int_{y_0}^{y} \frac{dy}{y} = \int_0^t -A\,dt$$

Then

$$\ln\frac{x}{x_0} = At \quad \text{and} \quad \ln\frac{y}{y_0} = -At$$

or

$$x = x_0 e^{At} \quad \text{and} \quad y = y_0 e^{-At}$$

At $t = 6$ s,

$$x = 2\,\text{m}\ e^{(0.3)6} = 12.1\,\text{m} \quad \text{and} \quad y = 8\,\text{m}\ e^{-(0.3)6} = 1.32\,\text{m}$$

At $t = 6$ s, particle is at (12.1, 1.32) m ←

(e) At the point (12.1, 1.32) m,

$$\vec{V} = A(x\hat{i} - y\hat{j}) = 0.3\,\text{s}^{-1}(12.1\hat{i} - 1.32\hat{j})\text{m}$$
$$= 3.63\hat{i} - 0.396\hat{j}\,\text{m/s} \longleftarrow$$

(f) To determine the equation of the pathline, we use the parametric equations

$$x = x_0 e^{At} \quad \text{and} \quad y = y_0 e^{-At}$$

and eliminate t. Solving for e^{At} from both equations

$$e^{At} = \frac{y_0}{y} = \frac{x}{x_0}$$

Therefore $xy = x_0 y_0 = 16\ \text{m}^2 \longleftarrow$

Notes:

- ✓ This problem illustrates the method for computing streamlines and pathlines.
- ✓ Because this is a steady flow, the streamlines and pathlines have the same shape—in an unsteady flow this would not be true.
- ✓ When we follow a particle (the Lagrangian approach), its position (x, y) and velocity ($u_p = dx/dt$ and $v_p = dy/dt$) are functions of time, even though the flow is steady.

Stress Field 2.3

In our study of fluid mechanics, we will need to understand what kinds of forces act on fluid particles. Each fluid particle can experience: *surface forces* (pressure, friction) that are generated by contact with other particles or a solid surface; and *body forces* (such as gravity and electromagnetic) that are experienced throughout the particle.

The gravitational body force acting on an element of volume, $d\forall$, is given by $\rho\vec{g}d\forall$, where ρ is the density (mass per unit volume) and $\vec{g}$ is the local gravitational acceleration. Thus the gravitational body force per unit volume is $\rho\vec{g}$ and the gravitational body force per unit mass is $\vec{g}$.

Surface forces on a fluid particle lead to *stresses*. The concept of stress is useful for describing how forces acting on the boundaries of a medium (fluid or solid) are transmitted throughout the medium. You have probably seen stresses discussed in solid mechanics. For example, when you stand on a diving board, stresses are generated within the board. On the other hand, when a body moves through a fluid, stresses are developed within the fluid. The difference between a fluid and a solid is, as we've seen, that stresses in a fluid are mostly generated by motion rather than by deflection.

Imagine the surface of a fluid particle in contact with other fluid particles, and consider the contact force being generated between the particles. Consider a portion, $\delta\vec{A}$, of the surface at some point C. The orientation of $\delta\vec{A}$ is given by the unit vector, $\hat{n}$, shown in Fig. 2.6. The vector $\hat{n}$ is the outwardly drawn unit normal with respect to the particle.

The force, $\delta\vec{F}$, acting on $\delta\vec{A}$ may be resolved into two components, one normal to and the other tangent to the area. A *normal stress* σ_n and a *shear stress* τ_n are then defined as

$$\sigma_n = \lim_{\delta A_n \to 0} \frac{\delta F_n}{\delta A_n} \tag{2.11}$$

and

$$\tau_n = \lim_{\delta A_n \to 0} \frac{\delta F_t}{\delta A_n} \tag{2.12}$$

Subscript n on the stress is included as a reminder that the stresses are associated with the surface $\delta\vec{A}$ through C, having an outward normal in the $\hat{n}$ direction. The fluid is

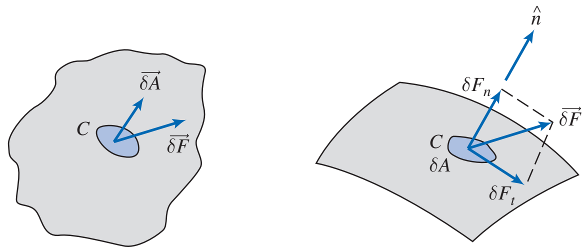


Fig. 2.6 The concept of stress in a continuum.

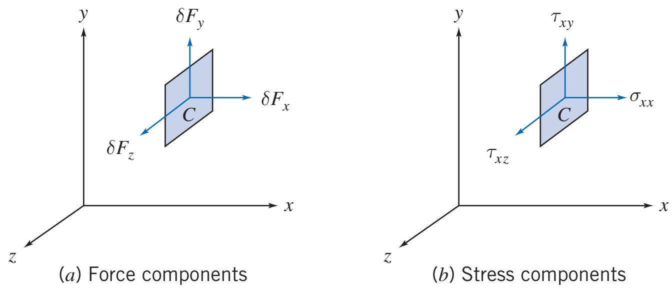


Fig. 2.7 Force and stress components on the element of area δA_x.

actually a continuum, so we could have imagined breaking it up any number of different ways into fluid particles around point C, and therefore obtained any number of different stresses at point C.

In dealing with vector quantities such as force, we usually consider components in an orthogonal coordinate system. In rectangular coordinates we might consider the stresses acting on planes whose outwardly drawn normals (again with respect to the material acted upon) are in the x, y, or z directions. In Fig. 2.7 we consider the stress on the element δA_x, whose outwardly drawn normal is in the x direction. The force, $\delta\vec{F}$, has been resolved into components along each of the coordinate directions. Dividing the magnitude of each force component by the area, δA_x, and taking the limit as δA_x approaches zero, we define the three stress components shown in Fig. 2.7b:

$$
\begin{aligned}
\sigma_{xx} &= \lim_{\delta A_x \to 0} \frac{\delta F_x}{\delta A_x} \\
\tau_{xy} &= \lim_{\delta A_x \to 0} \frac{\delta F_y}{\delta A_x} \qquad \tau_{xz} = \lim_{\delta A_x \to 0} \frac{\delta F_z}{\delta A_x}
\end{aligned}
\tag{2.13}
$$

We have used a double subscript notation to label the stresses. The *first* subscript (in this case, x) indicates the *plane* on which the stress acts (in this case, a surface perpendicular to the x axis). The *second* subscript indicates the *direction* in which the stress acts.

Consideration of area element δA_y would lead to the definitions of the stresses, σ_{yy}, τ_{yx}, and τ_{yz}; use of area element δA_z would similarly lead to the definitions of σ_{zz}, τ_{zx}, τ_{zy}.

Although we just looked at three orthogonal planes, an infinite number of planes can be passed through point C, resulting in an infinite number of stresses associated with planes through that point. Fortunately, the state of stress at a point can be described completely by specifying the stresses acting on *any* three mutually perpendicular planes through the point. The stress at a point is specified by the nine components

$$
\begin{bmatrix}
\sigma_{xx} & \tau_{xy} & \tau_{xz} \\
\tau_{yx} & \sigma_{yy} & \tau_{yz} \\
\tau_{zx} & \tau_{zy} & \sigma_{zz}
\end{bmatrix}
$$

where σ has been used to denote a normal stress, and τ to denote a shear stress. The notation for designating stress is shown in Fig. 2.8.

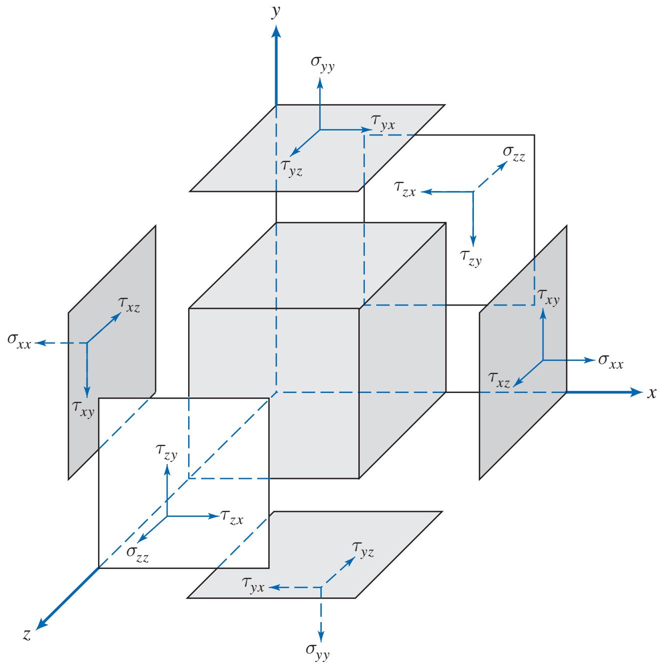


Fig. 2.8 Notation for stress.

Referring to the infinitesimal element shown in Fig. 2.8, we see that there are six planes (two x planes, two y planes, and two z planes) on which stresses may act. In order to designate the plane of interest, we could use terms like front and back, top and bottom, or left and right. However, it is more logical to name the planes in terms of the coordinate axes. The planes are named and denoted as positive or negative according to the direction of the outwardly drawn normal to the plane. Thus the top plane, for example, is a positive y plane and the back plane is a negative z plane.

It also is necessary to adopt a sign convention for stress. A stress component is positive when the direction of the stress component and the plane on which it acts are both positive or both negative. Thus $\tau_{yx} = 5\ \text{lbf/in.}^2$ represents a shear stress on a positive y plane in the positive x direction or a shear stress on a negative y plane in the negative x direction. In Fig. 2.8 all stresses have been drawn as positive stresses. Stress components are negative when the direction of the stress component and the plane on which it acts are of opposite sign.

Viscosity 2.4

Where do stresses come from? For a solid, stresses develop when the material is elastically deformed or strained; for a fluid, shear stresses arise due to viscous flow (we will discuss a fluid's normal stresses shortly). Hence we say solids are *elastic*, and fluids are *viscous* (and it's interesting to note that many biological tissues are *viscoelastic*, meaning they combine features of a solid and a fluid). For a fluid at rest, there will be no shear stresses. We will see that each fluid can be categorized by examining the relation between the applied shear stresses and the flow (specifically the rate of deformation) of the fluid.

Consider the behavior of a fluid element between the two infinite plates shown in Fig. 2.9*a*. The rectangular fluid element is initially at rest at time t. Let us now suppose a constant rightward force δF_x is applied to the upper plate so that it is dragged across the fluid at constant velocity δu. The relative shearing action of the infinite plates produces a shear stress, τ_{yx}, which acts on the fluid element and is given by

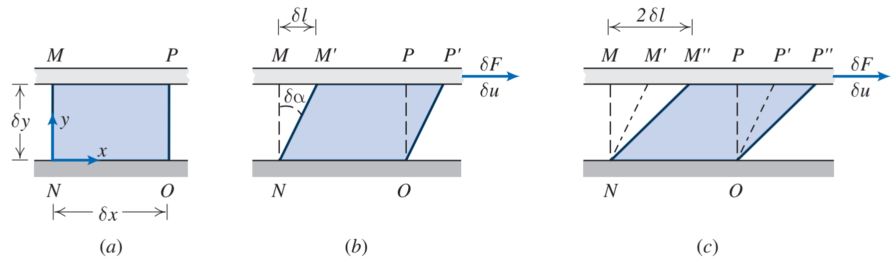


Fig. 2.9 (*a*) Fluid element at time t, (*b*) deformation of fluid element at time $t + \delta t$, and (*c*) deformation of fluid element at time $t + 2\delta t$.

$$\tau_{yx} = \lim_{\delta A_y \to 0} \frac{\delta F_x}{\delta A_y} = \frac{dF_x}{dA_y}$$

where δA_y is the area of contact of the fluid element with the plate and δF_x is the force exerted by the plate on that element. Snapshots of the fluid element, shown in Figs. 2.9*a–c*, illustrate the deformation of the fluid element from position $MNOP$ at time t, to $M'NOP'$ at time $t + \delta t$, to $M''NOP''$ at time $t + 2\delta t$, due to the imposed shear stress. As mentioned in Section 1.2, it is the fact that a fluid continually deforms in response to an applied shear stress that sets it apart from solids.

Focusing on the time interval δt (Fig. 2.9*b*), the deformation of the fluid is given by

$$\text{deformation rate} = \lim_{\delta t \to 0} \frac{\delta\alpha}{\delta t} = \frac{d\alpha}{dt}$$

We want to express $d\alpha/dt$ in terms of readily measurable quantities. This can be done easily. The distance, δl, between the points M and M' is given by

$$\delta l = \delta u\, \delta t$$

Alternatively, for small angles,

$$\delta l = \delta y\, \delta\alpha$$

Equating these two expressions for δl gives

$$\frac{\delta\alpha}{\delta t} = \frac{\delta u}{\delta y}$$

Taking the limits of both sides of the equality, we obtain

$$\frac{d\alpha}{dt} = \frac{du}{dy}$$

Thus, the fluid element of Fig. 2.9, when subjected to shear stress τ_{yx}, experiences a rate of deformation (*shear rate*) given by du/dy. We have established that any fluid that experiences a shear stress will flow (it will have a shear rate). What is the relation between shear stress and shear rate? Fluids in which shear stress is directly proportional to rate of deformation are *Newtonian fluids*. The term *non-Newtonian* is used to classify all fluids in which shear stress is not directly proportional to shear rate.

Newtonian Fluid

Most common fluids (the ones discussed in this text) such as water, air, and gasoline are Newtonian under normal conditions. If the fluid of Fig. 2.9 is Newtonian, then

$$\tau_{yx} \propto \frac{du}{dy} \tag{2.14}$$

We are familiar with the fact that some fluids resist motion more than others. For example, a container of SAE 30W oil is much harder to stir than one of water. Hence SAE 30W oil is much more viscous—it has a higher viscosity. (Note that a container of mercury is also harder to stir, but for a different reason!) The constant of

proportionality in Eq. 2.14 is the *absolute* (or *dynamic*) *viscosity*, μ. Thus in terms of the coordinates of Fig. 2.9, Newton's law of viscosity is given for one-dimensional flow by

$$\tau_{yx} = \mu \frac{du}{dy} \tag{2.15}$$

Note that, since the dimensions of τ are $[F/L^2]$ and the dimensions of du/dy are $[1/t]$, μ has dimensions $[Ft/L^2]$. Since the dimensions of force, F, mass, M, length, L, and time, t, are related by Newton's second law of motion, the dimensions of μ can also be expressed as $[M/Lt]$. In the British Gravitational system, the units of viscosity are $\text{lbf}\cdot\text{s/ft}^2$ or slug/(ft · s). In the Absolute Metric system, the basic unit of viscosity is called a poise [1 poise $\equiv$ 1 g/(cm · s)]; in the SI system the units of viscosity are kg/(m · s) or Pa · s (1 Pa · s = 1 N · s/m^2). The calculation of viscous shear stress is illustrated in Example 2.2.

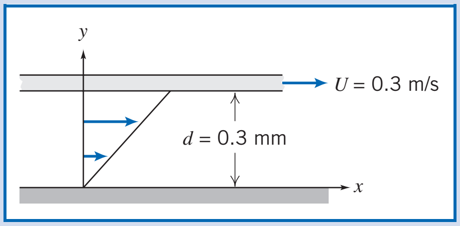
In fluid mechanics the ratio of absolute viscosity, μ, to density, ρ, often arises. This ratio is given the name *kinematic viscosity* and is represented by the symbol ν. Since density has dimensions $[M/L^3]$, the dimensions of ν are $[L^2/t]$. In the Absolute Metric system of units, the unit for ν is a stoke (1 stoke $\equiv$ 1 cm^2/s).

Viscosity data for a number of common Newtonian fluids are given in Appendix A. Note that for gases, viscosity increases with temperature, whereas for liquids, viscosity decreases with increasing temperature.

Example 2.2 VISCOSITY AND SHEAR STRESS IN NEWTONIAN FLUID

An infinite plate is moved over a second plate on a layer of liquid as shown. For small gap width, d, we assume a linear velocity distribution in the liquid. The liquid viscosity is 0.65 centipoise and its specific gravity is 0.88. Determine:

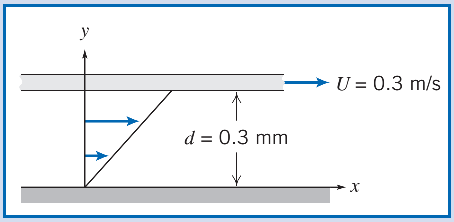
(a) The absolute viscosity of the liquid, in $\text{lbf}\cdot\text{s/ft}^2$.
(b) The kinematic viscosity of the liquid, in m^2/s.
(c) The shear stress on the upper plate, in lbf/ft^2.
(d) The shear stress on the lower plate, in Pa.
(e) The direction of each shear stress calculated in parts (c) and (d).



Given: Linear velocity profile in the liquid between infinite parallel plates as shown.

$$\mu = 0.65 \text{ cp}$$
$$\text{SG} = 0.88$$

Find: (a) μ in units of $\text{lbf}\cdot\text{s/ft}^2$.
(b) ν in units of m^2/s.
(c) τ on upper plate in units of lbf/ft^2.
(d) τ on lower plate in units of Pa.
(e) Direction of stresses in parts (c) and (d).



Solution:

Governing equation: $\tau_{yx} = \mu \dfrac{du}{dy}$ Definition: $\nu = \dfrac{\mu}{\rho}$

Assumptions: (1) Linear velocity distribution (given)
(2) Steady flow
(3) μ = constant

(a) $$\mu = 0.65\,\text{cp} \times \frac{\text{poise}}{100\,\text{cp}} \times \frac{\text{g}}{\text{cm}\cdot\text{s}\cdot\text{poise}} \times \frac{\text{lbm}}{454\,\text{g}} \times \frac{\text{slug}}{32.2\,\text{lbm}} \times 30.5\frac{\text{cm}}{\text{ft}} \times \frac{\text{lbf}\cdot\text{s}^2}{\text{slug}\cdot\text{ft}}$$

$$\mu = 1.36 \times 10^{-5}\ \text{lbf}\cdot\text{s/ft}^2 \longleftarrow \mu$$

(b) $\nu = \dfrac{\mu}{\rho} = \dfrac{\mu}{SG\ \rho_{H_2O}}$

$$= 1.36 \times 10^{-5} \frac{\text{lbf} \cdot \text{s}}{\text{ft}^2} \times \frac{\text{ft}^3}{(0.88)1.94\ \text{slug}} \times \frac{\text{slug} \cdot \text{ft}}{\text{lbf} \cdot \text{s}^2} \times (0.305)^2 \frac{\text{m}^2}{\text{ft}^2}$$

$$\nu = 7.41 \times 10^{-7} \text{m}^2/\text{s} \longleftarrow \nu$$

(c) $\tau_{\text{upper}} = \tau_{yx,\,\text{upper}} = \mu \left.\dfrac{du}{dy}\right)_{y=d}$

Since u varies linearly with y,

$$\frac{du}{dy} = \frac{\Delta u}{\Delta y} = \frac{U - 0}{d - 0} = \frac{U}{d}$$

$$= 0.3 \frac{\text{m}}{\text{s}} \times \frac{1}{0.3\ \text{mm}} \times 1000 \frac{\text{mm}}{\text{m}} = 1000\ \text{s}^{-1}$$

Part (c) shows that the shear stress is:

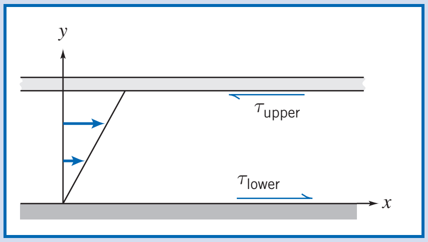
- ✓ Constant across the gap for a linear velocity profile.
- ✓ Directly proportional to the speed of the upper plate (because of the linearity of Newtonian fluids).
- ✓ Inversely proportional to the gap between the plates.

Note that multiplying the shear stress by the plate area in such problems computes the force required to maintain the motion.

$$\tau_{\text{upper}} = \mu \frac{U}{d} = 1.36 \times 10^{-5} \frac{\text{lbf} \cdot \text{s}}{\text{ft}^2} \times \frac{1000}{\text{s}} = 0.0136\ \text{lbf}/\text{ft}^2 \longleftarrow \tau_{\text{upper}}$$

(d) $\tau_{\text{lower}} = \mu \dfrac{U}{d} = 0.0136 \dfrac{\text{lbf}}{\text{ft}^2} \times 4.45 \dfrac{\text{N}}{\text{lbf}} \times \dfrac{\text{ft}^2}{(0.305)^2 \text{m}^2} \times \dfrac{\text{Pa} \cdot \text{m}^2}{\text{N}} = 0.651\ \text{Pa} \longleftarrow \tau_{\text{lower}}$

(e) Directions of shear stresses on upper and lower plates.



$\left\{ \begin{array}{l} \text{The upper plate is a negative } y \text{ surface; so} \\ \text{positive } \tau_{yx} \text{ acts in the negative } x \text{ direction.} \end{array} \right\}$

$\left\{ \begin{array}{l} \text{The lower plate is a positive } y \text{ surface; so} \\ \text{positive } \tau_{yx} \text{ acts in the positive } x \text{ direction.} \end{array} \right\}$

$\longleftarrow (e)$

Non-Newtonian Fluids

Fluids in which shear stress is *not* directly proportional to deformation rate are non-Newtonian. Although we will not discuss these much in this text, many common fluids exhibit non-Newtonian behavior. Two familiar examples are toothpaste and Lucite[5] paint. The latter is very "thick" when in the can, but becomes "thin" when sheared by brushing. Toothpaste behaves as a "fluid" when squeezed from the tube. However, it does not run out by itself when the cap is removed. There is a threshold or yield stress below which toothpaste behaves as a solid. Strictly speaking, our definition of a fluid is valid only for materials that have zero yield stress. Non-Newtonian fluids commonly are classified as having time-independent or time-dependent behavior. Examples of time-independent behavior are shown in the rheological diagram of Fig. 2.10.

Numerous empirical equations have been proposed [3, 4] to model the observed relations between τ_{yx} and du/dy for time-independent fluids. They may be adequately

[5]Trademark, E. I. du Pont de Nemours & Company.

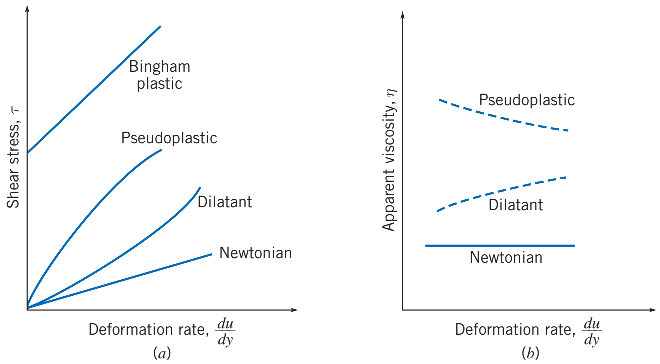


Fig. 2.10 (a) Shear stress, τ, and (b) apparent viscosity, η, as a function of deformation rate for one-dimensional flow of various non-Newtonian fluids.

represented for many engineering applications by the power law model, which for one-dimensional flow becomes

$$\tau_{yx} = k\left(\frac{du}{dy}\right)^n \tag{2.16}$$

where the exponent, n, is called the flow behavior index and the coefficient, k, the consistency index. This equation reduces to Newton's law of viscosity for $n = 1$ with $k = \mu$.

To ensure that τ_{yx} has the same sign as du/dy, Eq. 2.16 is rewritten in the form

$$\tau_{yx} = k\left|\frac{du}{dy}\right|^{n-1}\frac{du}{dy} = \eta\frac{du}{dy} \tag{2.17}$$

The term $\eta = k|du/dy|^{n-1}$ is referred to as the *apparent viscosity*. The idea behind Eq. 2.17 is that we end up with a viscosity η that is used in a formula that is the same form as Eq. 2.15, in which the Newtonian viscosity μ is used. The big difference is that while μ is constant (except for temperature effects), η depends on the shear rate. Most non-Newtonian fluids have apparent viscosities that are relatively high compared with the viscosity of water.

Fluids in which the apparent viscosity decreases with increasing deformation rate ($n < 1$) are called *pseudoplastic* (or shear thinning) fluids. Most non-Newtonian fluids fall into this group; examples include polymer solutions, colloidal suspensions, and paper pulp in water. If the apparent viscosity increases with increasing deformation rate ($n > 1$) the fluid is termed *dilatant* (or shear thickening). Suspensions of starch and of sand are examples of dilatant fluids. You can get an idea of the latter when you're on the beach—if you walk slowly (and hence generate a low shear rate) on very wet sand, you sink into it, but if you jog on it (generating a high shear rate), it's very firm.

A "fluid" that behaves as a solid until a minimum yield stress, τ_y, is exceeded and subsequently exhibits a linear relation between stress and rate of deformation is referred to as an ideal or *Bingham plastic*. The corresponding shear stress model is

$$\tau_{yx} = \tau_y + \mu_p\frac{du}{dy} \tag{2.18}$$

CLASSIC VIDEO

Rheological Behavior of Fluids.

Clay suspensions, drilling muds, and toothpaste are examples of substances exhibiting this behavior.

The study of non-Newtonian fluids is further complicated by the fact that the apparent viscosity may be time-dependent. *Thixotropic* fluids show a decrease in η with time under a constant applied shear stress; many paints are thixotropic. *Rheopectic* fluids show an increase in η with time. After deformation some fluids partially return to their original shape when the applied stress is released; such fluids are called *viscoelastic* (many biological fluids work this way).

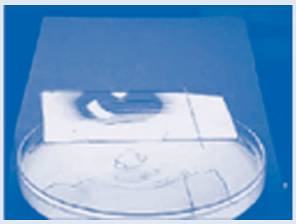
2.5 Surface Tension

VIDEO

Molecular Interactions at the Interface.

VIDEO

Shrinking Soap Film.



VIDEO

Soap Film Burst.

VIDEO

Wetted and Non-wetted Surfaces.

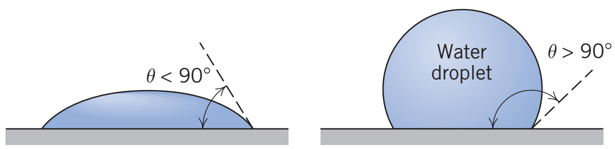
You can tell when your car needs waxing: Water droplets tend to appear somewhat flattened out. After waxing, you get a nice "beading" effect. These two cases are shown in Fig. 2.11. We define a liquid as "wetting" a surface when the *contact angle* $\theta < 90°$. By this definition, the car's surface was wetted before waxing, and not wetted after. This is an example of effects due to *surface tension*. Whenever a liquid is in contact with other liquids or gases, or in this case a gas/solid surface, an interface develops that acts like a stretched elastic membrane, creating surface tension. There are two features to this membrane: the contact angle, θ, and the magnitude of the surface tension, σ (N/m or lbf/ft). Both of these depend on the type of liquid and the type of solid surface (or other liquid or gas) with which it shares an interface. In the car-waxing example, the contact angle changed from being smaller than 90°, to larger than 90°, because, in effect, the waxing changed the nature of the solid surface. Factors that affect the contact angle include the cleanliness of the surface and the purity of the liquid.

Other examples of surface tension effects arise when you are able to place a needle on a water surface and, similarly, when small water insects are able to walk on the surface of the water.

Appendix A contains data for surface tension and contact angle for common liquids in the presence of air and of water.

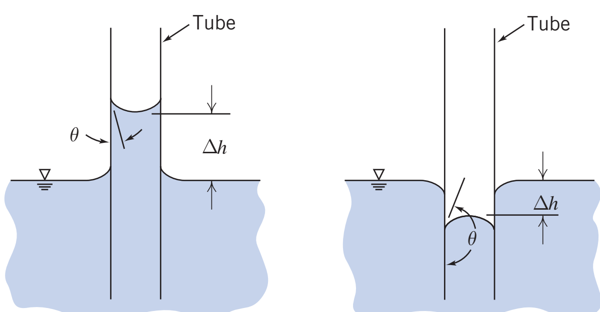
A force balance on a segment of interface shows that there is a pressure jump across the imagined elastic membrane whenever the interface is curved. For a water droplet in air, pressure in the water is higher than ambient; the same is true for a gas bubble in liquid. For a soap bubble in air, surface tension acts on both inside and outside interfaces between the soap film and air along the curved bubble surface. Surface tension also leads to the phenomena of capillary (i.e., very small wavelength) waves on a liquid surface [5], and capillary rise or depression, discussed below in Example 2.3.

In engineering, probably the most important effect of surface tension is the creation of a curved *meniscus* that appears in manometers or barometers, leading to a (usually unwanted) *capillary rise* (or depression), as shown in Fig. 2.12. This rise may be pronounced if the liquid is in a small-diameter tube or narrow gap, as shown in Example 2.3.



(*a*) A "wetted" surface　　(*b*) A nonwetted surface

Fig. 2.11 Surface tension effects on water droplets.

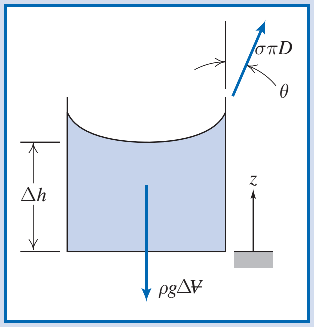


(*a*) Capillary rise ($\theta < 90°$)　　(*b*) Capillary depression ($\theta > 90°$)

Fig. 2.12 Capillary rise and capillary depression inside and outside a circular tube.

Example 2.3 ANALYSIS OF CAPILLARY EFFECT IN A TUBE

Create a graph showing the capillary rise or fall of a column of water or mercury, respectively, as a function of tube diameter D. Find the minimum diameter of each column required so that the height magnitude will be less than 1 mm.



Given: Tube dipped in liquid as in Fig. 2.12.

Find: A general expression for Δh as a function of D.

Solution:
Apply free-body diagram analysis, and sum vertical forces.

Governing equation:

$$\sum F_z = 0$$

Assumptions: (1) Measure to middle of meniscus
(2) Neglect volume in meniscus region

Summing forces in the z direction:

$$\sum F_z = \sigma \pi D \cos\theta - \rho g \Delta V\!\!\!\!/ = 0 \qquad (1)$$

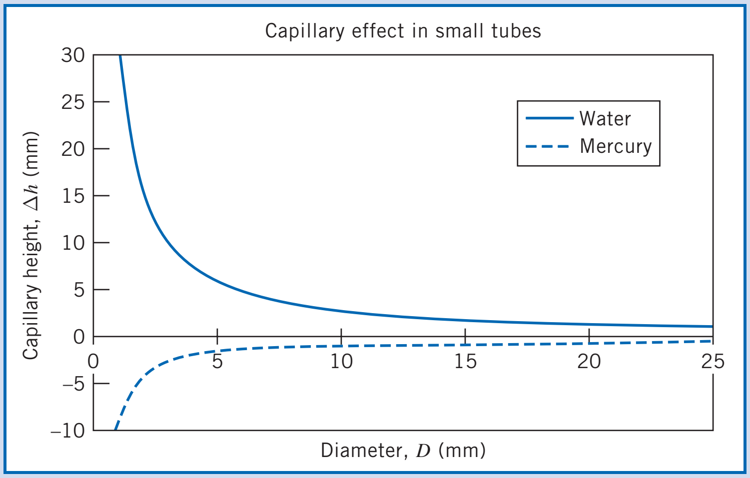
If we neglect the volume in the meniscus region:

$$\Delta V\!\!\!\!/ \approx \frac{\pi D^2}{4} \Delta h$$

Substituting in Eq. (1) and solving for Δh gives

$$\Delta h = \frac{4\sigma \cos\theta}{\rho g D} \longleftarrow \underline{\qquad \Delta h}$$

For water, $\sigma = 72.8$ mN/m and $\theta \approx 0°$, and for mercury, $\sigma = 484$ mN/m and $\theta = 140°$ (Table A.4). Plotting,



Using the above equation to compute $D_{\min}$ for $\Delta h = 1$ mm, we find for mercury and water

$$D_{M_{\min}} = 11.2 \text{ mm} \quad \text{and} \quad D_{W_{\min}} = 30 \text{ mm}$$

Notes:

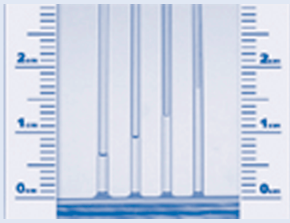
✓ This problem reviewed use of the free-body diagram approach.

✓ It turns out that neglecting the volume in the meniscus region is only valid when Δh is large compared with D. However, in this problem we have the result that Δh is about 1 mm when D is 11.2 mm (or 30 mm); hence the results can only be very approximate.

The graph and results were generated from the *Excel* workbook.

VIDEO

Capillary Rise.



Folsom [6] shows that the simple analysis of Example 2.3 overpredicts the capillary effect and gives reasonable results only for tube diameters less than 0.1 in. (2.54 mm). Over a diameter range $0.1 < D < 1.1$ in., experimental data for the capillary rise with a water-air interface are correlated by the empirical expression $\Delta h = 0.400/e^{4.37D}$.

Manometer and barometer readings should be made at the level of the middle of the meniscus. This is away from the maximum effects of surface tension and thus nearest to the proper liquid level.

All surface tension data in Appendix A were measured for pure liquids in contact with clean vertical surfaces. Impurities in the liquid, dirt on the surface, or surface inclination can cause an indistinct meniscus; under such conditions it may be difficult to determine liquid level accurately. Liquid level is most distinct in a vertical tube. When inclined tubes are used to increase manometer sensitivity (see Section 3.3) it is important to make each reading at the same point on the meniscus and to avoid use of tubes inclined less than about 15° from horizontal.

CLASSIC VIDEO

Surface Tension in Fluid Mechanics.

Surfactant compounds reduce surface tension significantly (more than 40% with little change in other properties [7]) when added to water. They have wide commercial application: Most detergents contain surfactants to help water penetrate and lift soil from surfaces. Surfactants also have major industrial applications in catalysis, aerosols, and oil field recovery.

2.6 Description and Classification of Fluid Motions

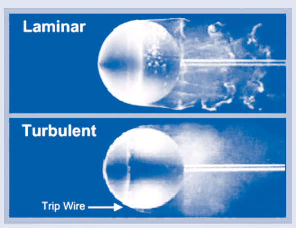
In Chapter 1 and in this chapter, we have almost completed our brief introduction to some concepts and ideas that are often needed when studying fluid mechanics. Before beginning detailed analysis of fluid mechanics in the rest of this text, we will describe some interesting examples to illustrate a broad classification of fluid mechanics on the basis of important flow characteristics. Fluid mechanics is a huge discipline: It covers everything from the aerodynamics of a supersonic transport vehicle to the lubrication of human joints by sinovial fluid. We need to break fluid mechanics down into manageable proportions. It turns out that the two most difficult aspects of a fluid mechanics analysis to deal with are: (1) the fluid's viscous nature and (2) its compressibility. In fact, the area of fluid mechanics theory that first became highly developed (about 250 years ago!) was that dealing with a frictionless, incompressible fluid. As we will see shortly (and in more detail later on), this theory, while extremely elegant, led to the famous result called d'Alembert's paradox: All bodies experience no drag as they move through such a fluid—a result not exactly consistent with any real behavior!

Although not the only way to do so, most engineers subdivide fluid mechanics in terms of whether or not viscous effects and compressibility effects are present, as shown in Fig. 2.13. Also shown are classifications in terms of whether a flow is laminar or turbulent, and internal or external. We will now discuss each of these.

Viscous and Inviscid Flows

VIDEO

Examples of Flow over a Sphere.



When you send a ball flying through the air (as in a game of baseball, soccer, or any number of other sports), in addition to gravity the ball experiences the aerodynamic drag of the air. The question arises: What is the nature of the drag force of the air on the ball? At first glance, we might conclude that it's due to friction of the air as it flows over the ball; a little more reflection might lead to the conclusion that because air has such a low viscosity, friction might not contribute much to the drag, and the drag might be due to the pressure build-up in front of the ball as it pushes the air out of the way. The question arises: Can we predict ahead of time the relative importance of the viscous force, and force due to the pressure build-up in front of the ball? Can we make similar predictions for *any* object, for example, an automobile, a submarine, a

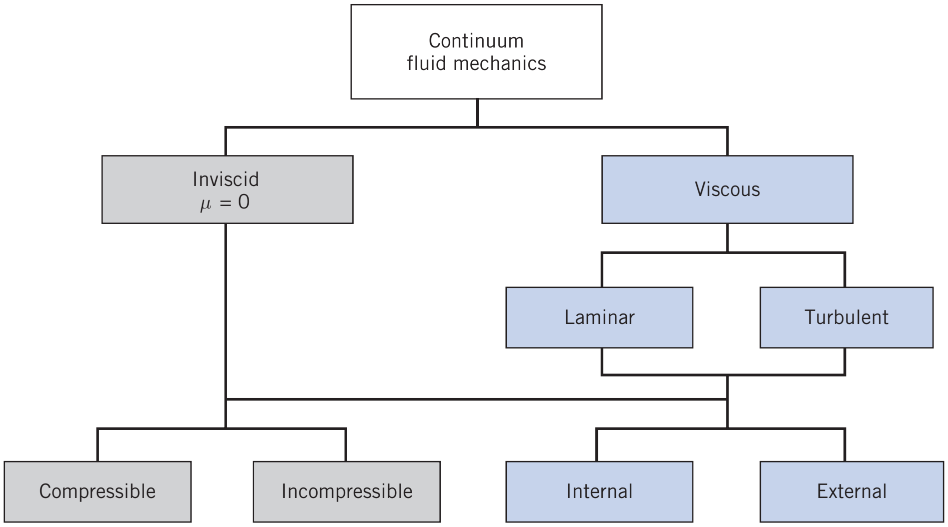


Fig. 2.13 Possible classification of continuum fluid mechanics.

VIDEO

Internal Laminar Flow in a Tube.

red blood cell, moving through *any* fluid, for example, air, water, blood plasma? The answer (which we'll discuss in much more detail in Chapter 7) is that we can! It turns out that we can estimate whether or not viscous forces, as opposed to pressure forces, are negligible by simply computing the Reynolds number

$$Re = \rho \frac{VL}{\mu}$$

where ρ and μ are the fluid density and viscosity, respectively, and V and L are the typical or "characteristic" velocity and size scale of the flow (in this example the ball velocity and diameter), respectively. If the Reynolds number is "large," viscous effects will be negligible (but will still have important consequences, as we'll soon see), at least in most of the flow; if the Reynolds number is small, viscous effects will be dominant. Finally, if the Reynolds number is neither large nor small, no general conclusions can be drawn.

To illustrate this very powerful idea, consider two simple examples. First, the drag on your ball: Suppose you kick a soccer ball (diameter = 8.75 in.) so it moves at 60 mph. The Reynolds number (using air properties from Table A.10) for this case is about 400,000—by any measure a large number; hence the drag on the soccer ball is almost entirely due to the pressure build-up in front of it. For our second example, consider a dust particle (modeled as a sphere of diameter 1 mm) falling under gravity at a terminal velocity of 1 cm/s: In this case $Re \approx 0.7$—a quite small number; hence the drag is mostly due to the friction of the air. Of course, in both of these examples, if we wish to *determine* the drag force, we would have to do substantially more analysis.

VIDEO

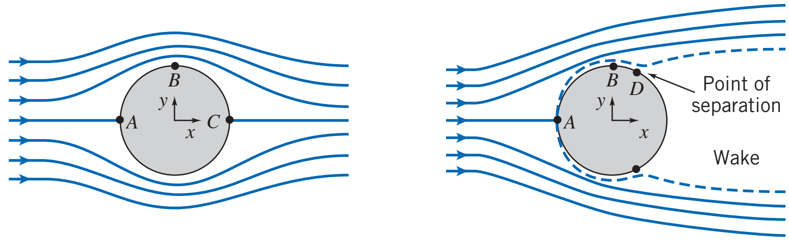
The Space Shuttle: An External Turbulent Flow.

These examples illustrate an important point: A flow is considered to be friction dominated (or not) based not just on the fluid's viscosity, but on the complete flow system. In these examples, the airflow was low friction for the soccer ball, but was high friction for the dust particle.

CLASSIC VIDEO

Fundamentals—Boundary Layers.

Let's return for a moment to the idealized notion of frictionless flow, called *inviscid flow*. This is the branch shown on the left in Fig. 2.13. This branch encompasses most aerodynamics, and among other things explains, for example, why sub- and supersonic aircraft have differing shapes, how a wing generates lift, and so forth. If this theory is applied to the ball flying through the air (a flow that is also incompressible), it predicts streamlines (in coordinates attached to the sphere) as shown in Fig. 2.14*a*.



(*a*) Inviscid flow (*b*) Viscous flow

Fig. 2.14 Qualitative picture of incompressible flow over a sphere.

The streamlines are symmetric front-to-back. Because the mass flow between any two streamlines is constant, wherever streamlines open up, the velocity must decrease, and vice versa. Hence we can see that the velocity in the vicinity of points A and C must be relatively low; at point B it will be high. In fact, the air comes to rest at points A and C: They are *stagnation points*. It turns out that (as we'll learn in Chapter 6) the pressure in this flow is high wherever the velocity is low, and vice versa. Hence, points A and C have relatively large (and equal) pressures; point B will be a point of low pressure. In fact, the pressure distribution on the sphere is symmetric front-to-back, and there is no net drag force due to pressure. Because we're assuming inviscid flow, there can be no drag due to friction either. Hence we have d'Alembert's paradox of 1752: The ball experiences no drag!

VIDEO

Boundary Layer Flow.

This is obviously unrealistic. On the other hand, everything seems logically consistent: We established that *Re* for the sphere was very large (400,000), indicating friction is negligible. We then used inviscid flow theory to obtain our no-drag result. How can we reconcile this theory with reality? It took about 150 years after the paradox first appeared for the answer, obtained by Prandtl in 1904: The no-slip condition (Section 1.2) requires that the velocity everywhere on the surface of the sphere be zero (in sphere coordinates), but inviscid theory states that it's high at point B. Prandtl suggested that even though friction is negligible in general for high-Reynolds number flows, there will always be a thin *boundary layer*, in which friction is significant and across the width of which the velocity increases rapidly from zero (at the surface) to the value inviscid flow theory predicts (on the outer edge of the boundary layer). This is shown in Fig. 2.14*b* from point A to point B, and in more detail in Fig. 2.15.

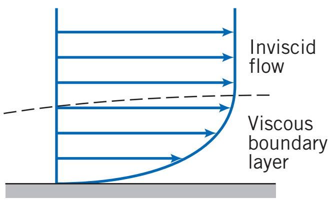


Fig. 2.15 Schematic of a boundary layer.

This boundary layer immediately allows us to reconcile theory and experiment: Once we have friction in a boundary layer we will have drag. However, this boundary layer has another important consequence: It often leads to bodies having a *wake*, as shown in Fig. 2.14*b* from point D onwards. Point D is a *separation point*, where fluid particles are pushed off the object and cause a wake to develop. Consider once again the original inviscid flow (Fig. 2.14*a*): As a particle moves along the surface from point B to C, it moves from low to high pressure. This *adverse pressure gradient* (a pressure change opposing fluid motion) causes the particles to slow down as they move along the rear of the sphere. If we now add to this the fact that the particles are moving in a boundary layer with friction that also slows down the fluid, the particles will eventually be brought to rest and then pushed off the sphere by the following particles, forming the wake. This is generally very bad news: It turns out that the wake will always be relatively low pressure, but the front of the sphere will still have relatively high pressure. Hence, the sphere will now have a quite large *pressure drag* (or *form drag*—so called because it's due to the shape of the object).

This description reconciles the inviscid flow no-drag result with the experimental result of significant drag on a sphere. It's interesting to note that although the boundary layer is necessary to explain the drag on the sphere, the drag is actually due

mostly to the asymmetric pressure distribution created by the boundary layer separation—drag directly due to friction is still negligible!

We can also now begin to see how *streamlining* of a body works. The drag force in most aerodynamics is due to the low-pressure wake: If we can reduce or eliminate the wake, drag will be greatly reduced. If we consider once again why the separation occurred, we recall two features: Boundary layer friction slowed down the particles, but so did the adverse pressure gradient. The pressure increased very rapidly across the back half of the sphere in Fig. 2.14*a* because the streamlines opened up so rapidly. If we make the sphere teardrop shaped, as in Fig. 2.16, the streamlines open up gradually, and hence the pressure will increase slowly, to such an extent that fluid particles are not forced to separate from the object until they almost reach the end of the object, as shown. The wake is much smaller (and it turns out the pressure will not be as low as before), leading to much less pressure drag. The only negative aspect of this streamlining is that the total surface area on which friction occurs is larger, so drag due to friction will increase a little.

We should point out that none of this discussion applies to the example of a falling dust particle: This low-Reynolds number flow was viscous throughout—there is no inviscid region.

Finally, this discussion illustrates the very significant difference between inviscid flow ($\mu = 0$) and flows in which viscosity is negligible but not zero ($\mu \to 0$).

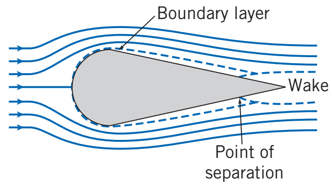
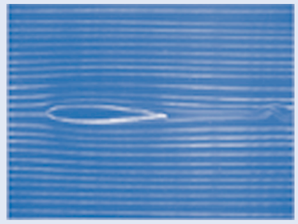


Fig. 2.16 Flow over a streamlined object.

VIDEO

Streamlined Flow over an Airfoil.



Laminar and Turbulent Flows

If you turn on a faucet (that doesn't have an aerator or other attachment) at a very low flow rate the water will flow out very smoothly—almost "glass-like." If you increase the flow rate, the water will exit in a churned-up, chaotic manner. These are examples of how a viscous flow can be laminar or turbulent, respectively. A *laminar* flow is one in which the fluid particles move in smooth layers, or laminas; a *turbulent* flow is one in which the fluid particles rapidly mix as they move along due to random three-dimensional velocity fluctuations. Typical examples of pathlines of each of these are illustrated in Fig. 2.17, which shows a one-dimensional flow. In most fluid mechanics problems—for example, flow of water in a pipe—turbulence is an unwanted but often unavoidable phenomenon, because it generates more resistance to flow; in other problems—for example, the flow of blood through blood vessels—it is desirable because the random mixing allows all of the blood cells to contact the walls of the blood vessels to exchange oxygen and other nutrients.

The velocity of the laminar flow is simply u; the velocity of the turbulent flow is given by the mean velocity $\bar{u}$ plus the three components of randomly fluctuating velocity u', v', and w'.

Although many turbulent flows of interest are steady in the mean ($\bar{u}$ is not a function of time), the presence of the random, high-frequency velocity fluctuations makes the analysis of turbulent flows extremely difficult. In a one-dimensional laminar flow, the shear stress is related to the velocity gradient by the simple relation

$$\tau_{yx} = \mu \frac{du}{dy} \tag{2.15}$$

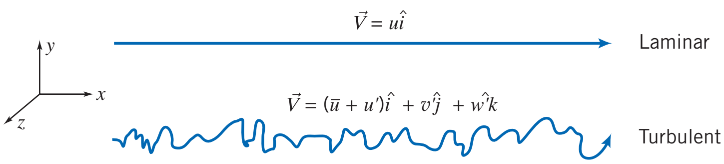


Fig. 2.17 Particle pathlines in one-dimensional laminar and turbulent flows.

VIDEO

Streamlines around a Car.

CLASSIC VIDEO

Fluid Dynamics of Drag, I–IV.

VIDEO

Laminar and Turbulent Flow.

For a turbulent flow in which the mean velocity field is one-dimensional, no such simple relation is valid. Random, three-dimensional velocity fluctuations (u', v', and w') transport momentum across the mean flow streamlines, increasing the effective shear stress. (This apparent stress is discussed in more detail in Chapter 8.) Consequently, in turbulent flow there is no universal relationship between the stress field and the mean-velocity field. Thus in turbulent flows we must rely heavily on semi-empirical theories and on experimental data.

Compressible and Incompressible Flows

Flows in which variations in density are negligible are termed *incompressible*; when density variations within a flow are not negligible, the flow is called *compressible*. The most common example of compressible flow concerns the flow of gases, while the flow of liquids may frequently be treated as incompressible.

For many liquids, density is only a weak function of temperature. At modest pressures, liquids may be considered incompressible. However, at high pressures, compressibility effects in liquids can be important. Pressure and density changes in liquids are related by the *bulk compressibility modulus*, or modulus of elasticity,

$$E_v \equiv \frac{dp}{(d\rho/\rho)} \tag{2.19}$$

If the bulk modulus is independent of temperature, then density is only a function of pressure (the fluid is *barotropic*). Bulk modulus data for some common liquids are given in Appendix A.

CLASSIC VIDEO

Cavitation.

Water hammer and cavitation are examples of the importance of compressibility effects in liquid flows. *Water hammer* is caused by acoustic waves propagating and reflecting in a confined liquid, for example, when a valve is closed abruptly. The resulting noise can be similar to "hammering" on the pipes, hence the term.

Cavitation occurs when vapor pockets form in a liquid flow because of local reductions in pressure (for example at the tip of a boat's propeller blades). Depending on the number and distribution of particles in the liquid to which very small pockets of undissolved gas or air may attach, the local pressure at the onset of cavitation may be at or below the vapor pressure of the liquid. These particles act as nucleation sites to initiate vaporization.

Vapor pressure of a liquid is the partial pressure of the vapor in contact with the saturated liquid at a given temperature. When pressure in a liquid is reduced to less than the vapor pressure, the liquid may change phase suddenly and "flash" to vapor.

The vapor pockets in a liquid flow may alter the geometry of the flow field substantially. When adjacent to a surface, the growth and collapse of vapor bubbles can cause serious damage by eroding the surface material.

Very pure liquids can sustain large negative pressures—as much as -60 atmospheres for distilled water—before the liquid "ruptures" and vaporization occurs. Undissolved air is invariably present near the free surface of water or seawater, so cavitation occurs where the local total pressure is quite close to the vapor pressure.

It turns out that gas flows with negligible heat transfer also may be considered incompressible provided that the flow speeds are small relative to the speed of sound; the ratio of the flow speed, V, to the local speed of sound, c, in the gas is defined as the Mach number,

$$M \equiv \frac{V}{c}$$

For $M < 0.3$, the maximum density variation is less than 5 percent. Thus gas flows with $M < 0.3$ can be treated as incompressible; a value of $M = 0.3$ in air at standard conditions corresponds to a speed of approximately 100 m/s. For example, although it might

be a little counterintuitive, when you drive your car at 65 mph the air flowing around it has negligible change in density. As we shall see in Chapter 12, the speed of sound in an ideal gas is given by $c = \sqrt{kRT}$, where k is the ratio of specific heats, R is the gas constant, and T is the absolute temperature. For air at STP, $k = 1.40$ and $R = 286.9$ J/kg·K (53.33 ft·lbf/lbm·°R). Values of k and R are supplied in Appendix A for several selected common gases at STP. In addition, Appendix A contains some useful data on atmospheric properties, such as temperature, at various elevations.

Compressible flows occur frequently in engineering applications. Common examples include compressed air systems used to power shop tools and dental drills, transmission of gases in pipelines at high pressure, and pneumatic or fluidic control and sensing systems. Compressibility effects are very important in the design of modern high-speed aircraft and missiles, power plants, fans, and compressors.

VIDEO

Compressible Flow: Shock Waves.

Internal and External Flows

Flows completely bounded by solid surfaces are called *internal* or *duct flows*. Flows over bodies immersed in an unbounded fluid are termed *external flows*. Both internal and external flows may be laminar or turbulent, compressible or incompressible.

We mentioned an example of internal flow when we discussed the flow out of a faucet—the flow in the pipe leading to the faucet is an internal flow. It turns out that we have a Reynolds number for pipe flows defined as $Re = \rho \overline{V} D/\mu$, where $\overline{V}$ is the average flow velocity and D is the pipe diameter (note that we do *not* use the pipe length!). This Reynolds number indicates whether a pipe flow will be laminar or turbulent. Flow will generally be laminar for $Re \le 2300$ and turbulent for larger values: Flow in a pipe of constant diameter will be entirely laminar or entirely turbulent, depending on the value of the velocity $\overline{V}$. We will explore internal flows in detail in Chapter 8.

We already saw some examples of external flows when we discussed the flow over a sphere (Fig. 2.14*b*) and a streamlined object (Fig. 2.16). What we didn't mention was that these flows could be laminar or turbulent. In addition, we mentioned boundary layers (Fig. 2.15): It turns out these also can be laminar or turbulent. When we discuss these in detail (Chapter 9), we'll start with the simplest kind of boundary layer—that over a flat plate—and learn that just as we have a Reynolds number for the overall external flow that indicates the relative significance of viscous forces, there will also be a boundary-layer Reynolds number $Re_x = \rho U_\infty x/\mu$ where in this case the characteristic velocity U_∞ is the velocity immediately outside the boundary layer and the characteristic length x is the distance along the plate. Hence, at the leading edge of the plate $Re_x = 0$, and at the end of a plate of length L, it will be $Re_x = \rho U_\infty L/\mu$. The significance of this Reynolds number is that (as we'll learn) the boundary layer will be laminar for $Re_x \le 5 \times 10^5$ and turbulent for larger values: A boundary layer will start out laminar, and if the plate is long enough the boundary layer will transition to become turbulent.

It is clear by now that computing a Reynolds number is often very informative for both internal and external flows. We will discuss this and other important *dimensionless groups* (such as the Mach number) in Chapter 7.

The internal flow through fluid machines is considered in Chapter 10. The principle of angular momentum is applied to develop fundamental equations for fluid machines. Pumps, fans, blowers, compressors, and propellers that add energy to fluid streams are considered, as are turbines and windmills that extract energy. The chapter features detailed discussion of operation of fluid systems.

The internal flow of liquids in which the duct does not flow full—where there is a free surface subject to a constant pressure—is termed *open-channel* flow. Common examples of open-channel flow include flow in rivers, irrigation ditches, and aqueducts. Open-channel flow will be treated in Chapter 11.

Both internal and external flows can be compressible or incompressible. Compressible flows can be divided into subsonic and supersonic regimes. We will study compressible flows in Chapters 12 and 13 and see among other things that *supersonic flows* ($M > 1$) will behave very differently than *subsonic flows* ($M < 1$). For example, supersonic flows can experience oblique and normal shocks, and can also behave in a counterintuitive way—e.g., a supersonic nozzle (a device to accelerate a flow) must be divergent (i.e., it has *increasing* cross-sectional area) in the direction of flow! We note here also that in a subsonic nozzle (which has a convergent cross-sectional area), the pressure of the flow at the exit plane will always be the ambient pressure; for a sonic flow, the exit pressure can be higher than ambient; and for a supersonic flow the exit pressure can be greater than, equal to, or less than the ambient pressure!

2.7 Summary and Useful Equations

In this chapter we have completed our review of some of the fundamental concepts we will utilize in our study of fluid mechanics. Some of these are:

- ✓ How to describe flows (timelines, pathlines, streamlines, streaklines).
- ✓ Forces (surface, body) and stresses (shear, normal).
- ✓ Types of fluids (Newtonian, non-Newtonian—dilatant, pseudoplastic, thixotropic, rheopectic, Bingham plastic) and viscosity (kinematic, dynamic, apparent).
- ✓ Types of flow (viscous/inviscid, laminar/turbulent, compressible/incompressible, internal/external).

We also briefly discussed some interesting phenomena, such as surface tension, boundary layers, wakes, and streamlining. Finally, we introduced two very useful dimensionless groups—the Reynolds number and the Mach number.

Note: Most of the Useful Equations in the table below have a number of constraints or limitations—*be sure to refer to their page numbers for details*!

Useful Equations

Definition of specific gravity:	$SG = \dfrac{\rho}{\rho_{H_2O}}$	**(2.3)**	Page 23
Definition of specific weight:	$\gamma = \dfrac{mg}{V\!\!\!\!-} \rightarrow \gamma = \rho g$	**(2.4)**	Page 23
Definition of streamlines (2D):	$\left.\dfrac{dy}{dx}\right)_{\text{streamline}} = \dfrac{v(x,y)}{u(x,y)}$	**(2.8)**	Page 27
Definition of pathlines (2D):	$\left.\dfrac{dx}{dt}\right)_{\text{particle}} = u(x,y,t) \qquad \left.\dfrac{dy}{dt}\right)_{\text{particle}} = v(x,y,t)$	**(2.9)**	Page 27
Definition of streaklines (2D):	$x_{\text{streakline}}(t_0) = x(t, x_0, y_0, t_0) \quad y_{\text{streakline}}(t_0) = y(t, x_0, y_0, t_0)$	**(2.10)**	Page 27
Newton's law of viscosity (1D flow):	$\tau_{yx} = \mu \dfrac{du}{dy}$	**(2.15)**	Page 33
Shear stress for a non-Newtonian fluid (1D flow):	$\tau_{yx} = k\left\lvert\dfrac{du}{dy}\right\rvert^{n-1}\dfrac{du}{dy} = \eta\dfrac{du}{dy}$	**(2.17)**	Page 35

Case Study

Fluid Mechanics and Your MP3 Player

The MP3 player of one of the authors.

Some people have the impression that fluid mechanics is old- or low-tech: water flow in a household pipe, the fluid forces acting on a dam, and so on. While it's true that many concepts in fluid mechanics are hundreds of years old, there are still lots of exciting new areas of research and development. Everyone has heard of the relatively high-tech area of fluid mechanics called streamlining (of cars, aircraft, racing bikes, and racing swimsuits, to mention a few), but there are many others.

If you're a typical engineering student, there's a decent chance that while reading this chapter you're listening to music on your MP3 player; you can thank fluid mechanics for your ability to do this! The tiny hard disk drive (HDD) in many of these devices typically holds about 250 gigabytes (GB) of data, so the disk platter must have a huge density (greater than 100,000 tracks per inch); in addition, the read/write head must get very close to the platter as it transfers data (typically the head is about 0.05 μm above the platter surface—a human hair is about 100 μm). The platter also spins at something greater than 500 revolutions per second! Hence the bearings in which the spindle of the platter spins must have very low friction but also have virtually no play or looseness—otherwise, at worst, the head will crash into the platter or, at best, you won't be able to read the data (it will be too closely packed). Designing such a bearing presents quite a challenge. Until a few years ago, most hard drives used ball bearings (BBs), which are essentially just like those in the wheel of a bicycle; they work on the principle that a spindle can rotate if it is held by a ring of small spheres that are supported in a cage. The problems with BBs are that they have a lot of components; they are very difficult to build to the precision needed for the HDD; they are vulnerable to shock (if you drop an HDD with such a drive, you're likely to dent one of the spheres as it hits the spindle, destroying the bearing); and they are relatively noisy.

Hard-drive makers are increasingly moving to fluid dynamic bearings (FDBs). These are mechanically much simpler than BBs; they consist basically of the spindle directly mounted in the bearing opening, with only a specially formulated viscous lubricant (such as ester oil) in the gap of only a few microns. The spindle and/or bearing surfaces have a herringbone pattern of grooves to maintain the oil in place. These bearings are extremely durable (they can often survive a shock of 500 g!) and low noise; they will also allow rotation speeds in excess of 15,000 rpm in the future, making data access even faster than with current devices. FDBs have been used before, in devices such as gyroscopes, but making them at such a small scale is new. Some FDBs even use pressurized air as the lubrication fluid, but one of the problems with these is that they sometimes stop working when you take them on an airplane flight—the cabin pressure is insufficient to maintain the pressure the bearing needs!

In recent times the price and capacity of flash memory have improved so much that many MP3 players are switching to this technology from HDDs. Eventually, notebook and desktop PCs will also switch to flash memory, but at least for the next few years HDDs will be the primary storage medium. Your PC will still have vital fluid-mechanical components!

References

1. Vincenti, W. G., and C. H. Kruger Jr., *Introduction to Physical Gas Dynamics*. New York: Wiley, 1965.
2. Merzkirch, W., *Flow Visualization*, 2nd ed. New York: Academic Press, 1987.
3. Tanner, R. I., *Engineering Rheology*. Oxford: Clarendon Press, 1985.
4. Macosko, C. W., *Rheology: Principles, Measurements, and Applications*. New York: VCH Publishers, 1994.
5. Loh, W. H. T., "Theory of the Hydraulic Analogy for Steady and Unsteady Gas Dynamics," in *Modern Developments in Gas Dynamics*, W. H. T. Loh, ed. New York: Plenum, 1969.
6. Folsom, R. G., "Manometer Errors Due to Capillarity," *Instruments*, 9, 1, 1937, pp. 36–37.
7. Waugh, J. G., and G. W. Stubstad, *Hydroballistics Modeling*. San Diego: Naval Undersea Center, ca. 1972.

Problems

Velocity Field

2.1 For the velocity fields given below, determine:
a. whether the flow field is one-, two-, or three-dimensional, and why.
b. whether the flow is steady or unsteady, and why.
(The quantities a and b are constants.)

(1) $\vec{V} = [(ax+t)e^{by}]\hat{i}$	(2) $\vec{V} = (ax - by)\hat{i}$
(3) $\vec{V} = ax\hat{i} + [e^{bx}]\hat{j}$	(4) $\vec{V} = ax\hat{i} + bx^2\hat{j} + ax\hat{k}$
(5) $\vec{V} = ax\hat{i} + [e^{bt}]\hat{j}$	(6) $\vec{V} = ax\hat{i} + bx^2\hat{j} + ay\hat{k}$
(7) $\vec{V} = ax\hat{i} + [e^{bt}]\hat{j} + ay\hat{k}$	(8) $\vec{V} = ax\hat{i} + [e^{by}]\hat{j} + az\hat{k}$

2.2 For the velocity fields given below, determine:
a. whether the flow field is one-, two-, or three-dimensional, and why.
b. whether the flow is steady or unsteady, and why.
(The quantities a and b are constants.)

(1) $\vec{V} = [ay^2e^{-bt}]\hat{i}$	(2) $\vec{V} = ax^2\hat{i} + bx\hat{j} + c\hat{k}$
(3) $\vec{V} = axy\hat{i} - byt\hat{j}$	(4) $\vec{V} = ax\hat{i} - by\hat{j} + ct\hat{k}$
(5) $\vec{V} = [ae^{-bx}]\hat{i} + bt^2\hat{j}$	(6) $\vec{V} = a(x^2+y^2)^{1/2}(1/z^3)\hat{k}$
(7) $\vec{V} = (ax+t)\hat{i} - by^2\hat{j}$	(8) $\vec{V} = ax^2\hat{i} + bxz\hat{j} + cy\hat{k}$

2.3 A viscous liquid is sheared between two parallel disks; the upper disk rotates and the lower one is fixed. The velocity field between the disks is given by $\vec{V} = \hat{e}_\theta r\omega z/h$. (The origin of coordinates is located at the center of the lower disk; the upper disk is located at $z = h$.) What are the dimensions of this velocity field? Does this velocity field satisfy appropriate physical boundary conditions? What are they?

2.4 For the velocity field $\vec{V} = Ax^2y\hat{i} + Bxy^2\hat{j}$, where $A = 2\ \text{m}^{-2}\text{s}^{-1}$ and $B = 1\ \text{m}^{-2}\text{s}^{-1}$, and the coordinates are measured in meters, obtain an equation for the flow streamlines. Plot several streamlines in the first quadrant.

2.5 The velocity field $\vec{V} = Ax\hat{i} - Ay\hat{j}$, where $A = 2\ \text{s}^{-1}$, can be interpreted to represent flow in a corner. Find an equation for the flow streamlines. Explain the relevance of A. Plot several streamlines in the first quadrant, including the one that passes through the point $(x, y) = (0, 0)$.

2.6 A velocity field is specified as $\vec{V} = axy\hat{i} + by^2\hat{j}$, where $a = 2\ \text{m}^{-1}\text{s}^{-1}$, $b = -6\ \text{m}^{-1}\text{s}^{-1}$, and the coordinates are measured in meters. Is the flow field one-, two-, or three-dimensional? Why? Calculate the velocity components at the point (2, ½). Develop an equation for the streamline passing through this point. Plot several streamlines in the first quadrant including the one that passes through the point (2, ½).

2.7 A velocity field is given by $\vec{V} = ax\hat{i} - bty\hat{j}$, where $a = 1\ \text{s}^{-1}$ and $b = 1\ \text{s}^{-2}$. Find the equation of the streamlines at any time t. Plot several streamlines in the first quadrant at $t = 0$ s, $t = 1$ s, and $t = 20$ s.

2.8 A velocity field is given by $\vec{V} = ax^3\hat{i} + bxy^3\hat{j}$, where $a = 1\ \text{m}^{-2}\text{s}^{-1}$ and $b = 1\ \text{m}^{-3}\text{s}^{-1}$. Find the equation of the streamlines. Plot several streamlines in the first quadrant.

2.9 A flow is described by the velocity field $\vec{V} = (Ax + B)\hat{i} + (-Ay)\hat{j}$, where $A = 10$ ft/s/ft and $B = 20$ ft/s. Plot a few streamlines in the xy plane, including the one that passes through the point $(x, y) = (1, 2)$.

2.10 The velocity for a steady, incompressible flow in the xy plane is given by $\vec{V} = \hat{i}A/x + \hat{j}Ay/x^2$, where $A = 2\ \text{m}^2/\text{s}$, and the coordinates are measured in meters. Obtain an equation for the streamline that passes through the point $(x, y) = (1, 3)$. Calculate the time required for a fluid particle to move from $x = 1$ m to $x = 2$ m in this flow field.

2.11 The flow field for an atmospheric flow is given by

$$\vec{V} = -\frac{My}{2\pi}\hat{i} + \frac{Mx}{2\pi}\hat{j}$$

where $M = 1\ \text{s}^{-1}$, and the x and y coordinates are the parallel to the local latitude and longitude. Plot the velocity magnitude along the x axis, along the y axis, and along the line $y = x$, and discuss the velocity direction with respect to these three axes. For each plot use a range x or $y = 0$ km to 1 km. Find the equation for the streamlines and sketch several of them. What does this flow field model?

2.12 The flow field for an atmospheric flow is given by

$$\vec{V} = -\frac{Ky}{2\pi(x^2+y^2)}\hat{i} + \frac{Kx}{2\pi(x^2+y^2)}\hat{j}$$

where $K = 10^5\ \text{m}^2/\text{s}$, and the x and y coordinates are parallel to the local latitude and longitude. Plot the velocity magnitude along the x axis, along the y axis, and along the line $y = x$, and discuss the velocity direction with respect to these three axes. For each plot use a range x or $y = -1$ km to 1 km, excluding $|x|$ or $|y| < 100$ m. Find the equation for the streamlines and sketch several of them. What does this flow field model?

2.13 A flow field is given by

$$\vec{V} = -\frac{qx}{2\pi(x^2+y^2)}\hat{i} - \frac{qy}{2\pi(x^2+y^2)}\hat{j}$$

where $q = 5 \times 10^4$ m^2/s. Plot the velocity magnitude along the x axis, along the y axis, and along the line $y = x$, and discuss the velocity direction with respect to these three axes. For each plot use a range x or $y = -1$ km to 1 km, excluding $|x|$ or $|y| < 100$ m. Find the equation for the streamlines and sketch several of them. What does this flow field model?

2.14 Beginning with the velocity field of Problem 2.5, show that the parametric equations for particle motion are given by $x_p = c_1 e^{At}$ and $y_p = c_2 e^{-At}$. Obtain the equation for the pathline of the particle located at the point $(x, y) = (2, 2)$ at the instant $t = 0$. Compare this pathline with the streamline through the same point.

2.15 A flow field is given by $\vec{V} = Ax\hat{i} + 2Ay\hat{j}$, where $A = 2\ \text{s}^{-1}$. Verify that the parametric equations for particle motion are given by $x_p = c_1 e^{At}$ and $y_p = c_2 e^{2At}$. Obtain the equation for the pathline of the particle located at the point $(x, y) = (2, 2)$ at the instant $t = 0$. Compare this pathline with the streamline through the same point.

2.16 A velocity field is given by $\vec{V} = ayt\hat{i} - bx\hat{j}$, where $a = 1\ \text{s}^{-2}$ and $b = 4\ \text{s}^{-1}$. Find the equation of the streamlines at any time t. Plot several streamlines at $t = 0$ s, $t = 1$ s, and $t = 20$ s.

2.17 Verify that $x_p = -a\sin(\omega t)$, $y_p = a\cos(\omega t)$ is the equation for the pathlines of particles for the flow field of Problem 2.12. Find the frequency of motion ω as a function of the amplitude of motion, a, and K. Verify that $x_p = -a\sin(\omega t)$, $y_p = a\cos(\omega t)$ is also the equation for the pathlines of particles for the flow field of Problem 2.11, except that ω is now a function of M. Plot typical pathlines for both flow fields and discuss the difference.

2.18 Air flows downward toward an infinitely wide horizontal flat plate. The velocity field is given by $\vec{V} = (ax\hat{i} - ay\hat{j})(2 + \cos\omega t)$, where $a = 5\ \text{s}^{-1}$, $\omega = 2\pi\ \text{s}^{-1}$, x and y (measured in meters) are horizontal and vertically upward, respectively, and t is in s. Obtain an algebraic equation for a streamline at $t = 0$. Plot the streamline that passes through point $(x, y) = (3, 3)$ at this instant. Will the streamline change with time? Explain briefly. Show the velocity vector on your plot at the same point and time. Is the velocity vector tangent to the streamline? Explain.

2.19 Consider the flow described by the velocity field $\vec{V} = A(1 + Bt)\hat{i} + Cty\hat{j}$, with $A = 1$ m/s, $B = 1\ \text{s}^{-1}$, and $C = 1\ \text{s}^{-2}$. Coordinates are measured in meters. Plot the pathline traced out by the particle that passes through the point $(1, 1)$ at time $t = 0$. Compare with the streamlines plotted through the same point at the instants $t = 0$, 1, and 2 s.

2.20 Consider the flow described by the velocity field $\vec{V} = Bx(1 + At)\hat{i} + Cy\hat{j}$, with $A = 0.5\ \text{s}^{-1}$ and $B = C = 1\ \text{s}^{-1}$. Coordinates are measured in meters. Plot the pathline traced out by the particle that passes through the point $(1, 1)$ at time $t = 0$. Compare with the streamlines plotted through the same point at the instants $t = 0$, 1, and 2 s.

2.21 Consider the flow field given in Eulerian description by the expression $\vec{V} = A\hat{i} - Bt\hat{j}$, where $A = 2$ m/s, $B = 2$ m/s^2, and the coordinates are measured in meters. Derive the Lagrangian position functions for the fluid particle that was located at the point $(x, y) = (1, 1)$ at the instant $t = 0$. Obtain an algebraic expression for the pathline followed by this particle. Plot the pathline and compare with the streamlines plotted through the same point at the instants $t = 0$, 1, and 2 s.

2.22 Consider the velocity field $V = ax\hat{i} + by(1 + ct)\hat{j}$, where $a = b = 2\ \text{s}^{-1}$ and $c = 0.4\ \text{s}^{-1}$. Coordinates are measured in meters. For the particle that passes through the point $(x, y) = (1, 1)$ at the instant $t = 0$, plot the pathline during the interval from $t = 0$ to 1.5 s. Compare this pathline with the streamlines plotted through the same point at the instants $t = 0$, 1, and 1.5 s.

2.23 Consider the flow field given in Eulerian descriptionby the expression $\vec{V} = ax\hat{i} + byt\hat{j}$, where $a = 0.2\ \text{s}^{-1}$, $b = 0.04\ \text{s}^{-2}$, and the coordinates are measured in meters. Derive the Lagrangian position functions for the fluid particle that was located at the point $(x, y) = (1, 1)$ at the instant $t = 0$. Obtain an algebraic expression for the pathline followed by this particle. Plot the pathline and compare with the streamlines plotted through the same point at the instants $t = 0$, 10, and 20 s.

2.24 A velocity field is given by $\vec{V} = axt\hat{i} - by\hat{j}$, where $a = 0.1\ \text{s}^{-2}$ and $b = 1\ \text{s}^{-1}$. For the particle that passes through the point $(x, y) = (1, 1)$ at instant $t = 0$ s, plot the pathline during the interval from $t = 0$ to $t = 3$ s. Compare with the streamlines plotted through the same point at the instants $t = 0$, 1, and 2 s.

2.25 Consider the flow field $\vec{V} = axt\hat{i} + b\hat{j}$, where $a = 0.1\ \text{s}^{-2}$ and $b = 4$ m/s. Coordinates are measured in meters. For the particle that passes through the point $(x, y) = (3, 1)$ at the instant $t = 0$, plot the pathline during the interval from $t = 0$ to 3 s. Compare this pathline with the streamlines plotted through the same point at the instants $t = 1$, 2, and 3 s.

2.26 Consider the garden hose of Fig. 2.5. Suppose the velocity field is given by $\vec{V} = u_0\hat{i} + v_0\sin[\omega(t - x/u_0)]\hat{j}$, where the x direction is horizontal and the origin is at the mean position of the hose, $u_0 = 10$ m/s, $v_0 = 2$ m/s, and $\omega = 5$ cycle/s. Find and plot on one graph the instantaneous streamlines that pass through the origin at $t = 0$ s, 0.05 s, 0.1 s, and 0.15 s. Also find and plot on one graph the pathlines of particles that left the origin at the same four times.

2.27 Using the data of Problem 2.26, find and plot the streakline shape produced after the first second of flow.

2.28 Consider the velocity field of Problem 2.20. Plot the streakline formed by particles that passed through the point $(1, 1)$ during the interval from $t = 0$ to $t = 3$ s. Compare with the streamlines plotted through the same point at the instants $t = 0$, 1, and 2 s.

2.29 Streaklines are traced out by neutrally buoyant marker fluid injected into a flow field from a fixed point in space. A particle of the marker fluid that is at point (x, y) at time t must have passed through the injection point (x_0, y_0) at some earlier instant $t = \tau$. The time history of a marker particle may be found by solving the pathline equations for the initial conditions that $x = x_0$, $y = y_0$ when $t = \tau$. The present locations of particles on the streakline are obtained by setting τ equal to values in the range $0 \le \tau \le t$. Consider the flow field $\vec{V} = ax(1 + bt)\hat{i} + cy\hat{j}$, where $a = c = 1\ \text{s}^{-1}$ and $b = 0.2\ \text{s}^{-1}$. Coordinates are measured in meters. Plot the streakline that

passes through the initial point $(x_0, y_0) = (1, 1)$, during the interval from $t = 0$ to $t = 3$ s. Compare with the streamline plotted through the same point at the instants $t = 0$, 1, and 2 s.

2.30 Consider the flow field $\vec{V} = axt\hat{i} + b\hat{j}$, where $a = 1/4\ \text{s}^{-2}$ and $b = 1/3$ m/s. Coordinates are measured in meters. For the particle that passes through the point $(x, y) = (1, 2)$ at the instant $t = 0$, plot the pathline during the time interval from $t = 0$ to 3 s. Compare this pathline with the streakline through the same point at the instant $t = 3$ s.

2.31 A flow is described by velocity field $\vec{V} = ay^2\hat{i} + b\hat{j}$, where $a = 1\ \text{m}^{-1}\text{s}^{-1}$ and $b = 2$ m/s. Coordinates are measured in meters. Obtain the equation for the streamline passing through point (6, 6). At $t = 1$ s, what are the coordinates of the particle that passed through point (1, 4) at $t = 0$? At $t = 3$ s, what are the coordinates of the particle that passed through point $(-3, 0)$ 2 s earlier? Show that pathlines, streamlines, and streaklines for this flow coincide.

2.32 Tiny hydrogen bubbles are being used as tracers to visualize a flow. All the bubbles are generated at the origin $(x = 0, y = 0)$. The velocity field is unsteady and obeys the equations:

$$\begin{array}{lll} u = 1\ \text{m/s} & v = 2\ \text{m/s} & 0 \le t < 2\ \text{s} \\ u = 0 & v = -1\ \text{m/s} & 0 \le t \le 4\ \text{s} \end{array}$$

Plot the pathlines of bubbles that leave the origin at $t = 0$, 1, 2, 3, and 4 s. Mark the locations of these five bubbles at $t = 4$ s. Use a dashed line to indicate the position of a streakline at $t = 4$ s.

2.33 A flow is described by velocity field $\vec{V} = ax\hat{i} + b\hat{j}$, where $a = 1/5\ \text{s}^{-1}$ and $b = 1$ m/s. Coordinates are measured in meters. Obtain the equation for the streamline passing through point (1, 1). At $t = 5$ s, what are the coordinates of the particle that initially (at $t = 0$) passed through point (1, 1)? What are its coordinates at $t = 10$ s? Plot the streamline and the initial, 5 s, and 10 s positions of the particle. What conclusions can you draw about the pathline, streamline, and streakline for this flow?

2.34 A flow is described by velocity field $\vec{V} = a\hat{i} + bx\hat{j}$, where $a = 2$ m/s and $b = 1\ \text{s}^{-1}$. Coordinates are measured in meters. Obtain the equation for the streamline passing through point (2, 5). At $t = 2$ s, what are the coordinates of the particle that passed through point (0, 4) at $t = 0$? At $t = 3$ s, what are the coordinates of the particle that passed through point (1, 4.25) 2 s earlier? What conclusions can you draw about the pathline, streamline, and streakline for this flow?

2.35 A flow is described by velocity field $\vec{V} = ay\hat{i} + bt\hat{j}$, where $a = 0.2\ \text{s}^{-1}$ and $b = 0.4\ \text{m/s}^2$. At $t = 2$ s, what are the coordinates of the particle that passed through point (1, 2) at $t = 0$? At $t = 3$ s, what are the coordinates of the particle that passed through point (1, 2) at $t = 2$ s? Plot the pathline and streakline through point (1, 2), and plot the streamlines through the same point at the instants $t = 0$, 1, 2, and 3 s.

2.36 A flow is described by velocity field $\vec{V} = at\hat{i} + b\hat{j}$, where $a = 0.4\ \text{m/s}^2$ and $b = 2$ m/s. At $t = 2$ s, what are the coordinates of the particle that passed through point (2, 1) at $t = 0$? At $t = 3$ s, what are the coordinates of the particle that passed through point (2, 1) at $t = 2$ s? Plot the pathline and streakline through point (2, 1) and compare with the streamlines through the same point at the instants $t = 0$, 1, and 2 s.

Viscosity

2.37 The variation with temperature of the viscosity of air is represented well by the empirical Sutherland correlation

$$\mu = \frac{bT^{1/2}}{1 + S/T}$$

Best-fit values of b and S are given in Appendix A. Develop an equation in SI units for kinematic viscosity versus temperature for air at atmospheric pressure. Assume ideal gas behavior. Check by using the equation to compute the kinematic viscosity of air at 0°C and at 100°C and comparing to the data in Appendix 10 (Table A.10); plot the kinematic viscosity for a temperature range of 0°C to 100°C, using the equation and the data in Table A.10.

2.38 The variation with temperature of the viscosity of air is correlated well by the empirical Sutherland equation

$$\mu = \frac{bT^{1/2}}{1 + S/T}$$

Best-fit values of b and S are given in Appendix A for use with SI units. Use these values to develop an equation for calculating air viscosity in British Gravitational units as a function of absolute temperature in degrees Rankine. Check your result using data from Appendix A.

2.39 Some experimental data for the viscosity of helium at 1 atm are

T, °C	0	100	200	300	400
μ, N·s/m^2($\times 10^5$)	1.86	2.31	2.72	3.11	3.46

Using the approach described in Appendix A.3, correlate these data to the empirical Sutherland equation

$$\mu = \frac{bT^{1/2}}{1 + S/T}$$

(where T is in kelvin) and obtain values for constants b and S.

2.40 The velocity distribution for laminar flow between parallel plates is given by

$$\frac{u}{u_{\max}} = 1 - \left(\frac{2y}{h}\right)^2$$

where h is the distance separating the plates and the origin is placed midway between the plates. Consider a flow of water at 15°C, with $u_{\max} = 0.10$ m/s and $h = 0.1$ mm. Calculate the shear stress on the upper plate and give its direction. Sketch the variation of shear stress across the channel.

2.41 The velocity distribution for laminar flow between parallel plates is given by

$$\frac{u}{u_{\max}} = 1 - \left(\frac{2y}{h}\right)^2$$

where h is the distance separating the plates and the origin is placed midway between the plates. Consider a flow of water at 15°C with maximum speed of 0.05 m/s and $h = 0.1$ mm.

Calculate the force on a 1 m^2 section of the lower plate and give its direction.

2.42 Explain how an ice skate interacts with the ice surface. What mechanism acts to reduce sliding friction between skate and ice?

2.43 Crude oil, with specific gravity SG = 0.85 and viscosity $\mu = 2.15 \times 10^{-3}$ $\text{lbf}\cdot\text{s/ft}^2$, flows steadily down a surface inclined $\theta = 45$ degrees below the horizontal in a film of thickness $h = 0.1$ in. The velocity profile is given by

$$u = \frac{\rho g}{\mu}\left(hy - \frac{y^2}{2}\right)\sin\theta$$

(Coordinate x is along the surface and y is normal to the surface.) Plot the velocity profile. Determine the magnitude and direction of the shear stress that acts on the surface.

2.44 A female freestyle ice skater, weighing 100 lbf, glides on one skate at speed $V = 20$ ft/s. Her weight is supported by a thin film of liquid water melted from the ice by the pressure of the skate blade. Assume the blade is $L = 11.5$ in. long and $w = 0.125$ in. wide, and that the water film is $h = 0.0000575$ in. thick. Estimate the deceleration of the skater that results from viscous shear in the water film, if end effects are neglected.

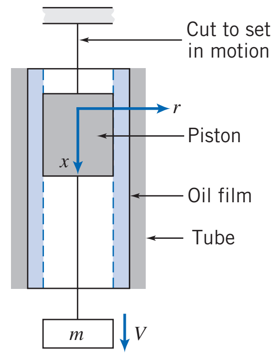
2.45 A block weighing 10 lbf and having dimensions 10 in. on each edge is pulled up an inclined surface on which there is a film of SAE 10W oil at 100°F. If the speed of the block is 2 ft/s and the oil film is 0.001 in. thick, find the force required to pull the block. Assume the velocity distribution in the oil film is linear. The surface is inclined at an angle of 25° from the horizontal.

2.46 A block of mass 10 kg and measuring 250 mm on each edge is pulled up an inclined surface on which there is a film of SAE 10W-30 oil at 30°F (the oil film is 0.025 mm thick). Find the steady speed of the block if it is released. If a force of 75 N is applied to pull the block up the incline, find the steady speed of the block. If the force is now applied to push the block down the incline, find the steady speed of the block. Assume the velocity distribution in the oil film is linear. The surface is inclined at an angle of 30° from the horizontal.

2.47 Tape is to be coated on both sides with glue by drawing it through a narrow gap. The tape is 0.015 in. thick and 1.00 in. wide. It is centered in the gap with a clearance of 0.012 in. on each side. The glue, of viscosity $\mu = 0.02$ slug/(ft·s), completely fills the space between the tape and gap. If the tape can withstand a maximum tensile force of 25 lbf, determine the maximum gap region through which it can be pulled at a speed of 3 ft/s.

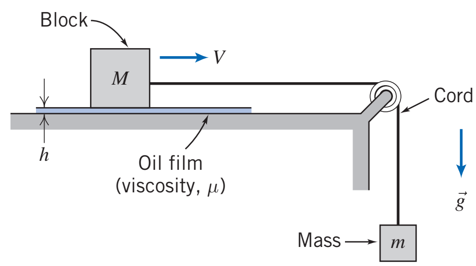
2.48 A 73-mm-diameter aluminum (SG = 2.64) piston of 100-mm length resides in a stationary 75-mm-inner-diameter steel tube lined with SAE 10W-30 oil at 25°C. A mass $m = 2$ kg is suspended from the free end of the piston. The piston is set into motion by cutting a support cord. What is the terminal velocity of mass m? Assume a linear velocity profile within the oil.

2.49 The piston in Problem 2.48 is traveling at terminal speed. The mass m now disconnects from the piston. Plot the piston speed vs. time. How long does it take the piston to come within 1 percent of its new terminal speed?



P2.48, P2.49

2.50 A block of mass M slides on a thin film of oil. The film thickness is h and the area of the block is A. When released, mass m exerts tension on the cord, causing the block to accelerate. Neglect friction in the pulley and air resistance. Develop an algebraic expression for the viscous force that acts on the block when it moves at speed V. Derive a differential equation for the block speed as a function of time. Obtain an expression for the block speed as a function of time. The mass $M = 5$ kg, $m = 1$ kg, $A = 25$ cm^2, and $h = 0.5$ mm. If it takes 1 s for the speed to reach 1 m/s, find the oil viscosity μ. Plot the curve for $V(t)$.



P2.50

2.51 A block 0.1 m square, with 5 kg mass, slides down a smooth incline, 30° below the horizontal, on a film of SAE 30 oil at 20°C that is 0.20 mm thick. If the block is released from rest at $t = 0$, what is its initial acceleration? Derive an expression for the speed of the block as a function of time. Plot the curve for $V(t)$. Find the speed after 0.1 s. If we want the mass to instead reach a speed of 0.3 m/s at this time, find the viscosity μ of the oil we would have to use.

2.52 A block that is a mm square slides across a flat plate on a thin film of oil. The oil has viscosity μ and the film is h mm thick. The block of mass M moves at steady speed U under the influence of constant force F. Indicate the magnitude and direction of the shear stresses on the bottom of the block and the plate. If the force is removed suddenly and the block begins to slow, sketch the resulting speed versus time curve for the block. Obtain an expression for the time required for the block to lose 95 percent of its initial speed.

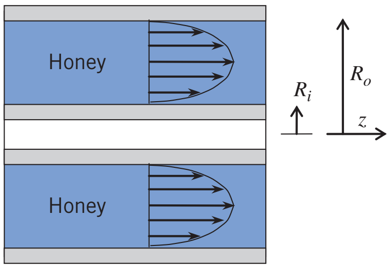
2.53 Magnet wire is to be coated with varnish for insulation by drawing it through a circular die of 1.0 mm diameter. The wire diameter is 0.9 mm and it is centered in the die.

The varnish ($\mu = 20$ centipoise) completely fills the space between the wire and the die for a length of 50 mm. The wire is drawn through the die at a speed of 50 m/s. Determine the force required to pull the wire.

2.54 In a food-processing plant, honey is pumped through an annular tube. The tube is $L = 2$ m long, with inner and outer radii of $R_i = 5$ mm and $R_o = 25$ mm, respectively. The applied pressure difference is $\Delta p = 125$ kPa, and the honey viscosity is $\mu = 5\ \mathrm{N \cdot s/m^2}$. The theoretical velocity profile for laminar flow through an annulus is:

$$u_z(r) = \frac{1}{4\mu}\left(\frac{\Delta p}{L}\right)\left[R_i^2 - r^2 - \frac{R_o^2 - R_i^2}{\ln\left(\frac{R_i}{R_o}\right)} \cdot \ln\left(\frac{r}{R_i}\right)\right]$$

Show that the no-slip condition is satisfied by this expression. Find the location at which the shear stress is zero. Find the viscous forces acting on the inner and outer surfaces, and compare these to the force $\Delta p \pi (R_o^2 - R_i^2)$. Explain.



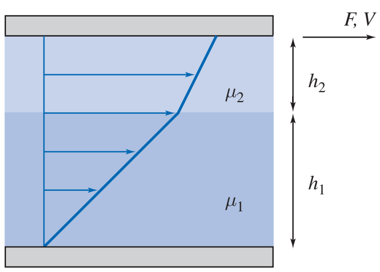
P2.54

2.55 SAE 10W-30 oil at 100°C is pumped through a tube $L = 10$ m long, diameter $D = 20$ mm. The applied pressure difference is $\Delta p = 5$ kPa. On the centerline of the tube is a metal filament of diameter $d = 1\ \mu$m. The theoretical velocity profile for laminar flow through the tube is:

$$V(r) = \frac{1}{16\mu}\left(\frac{\Delta p}{L}\right)\left[d^2 - 4r^2 - \frac{D^2 - d^2}{\ln\left(\frac{d}{D}\right)} \cdot \ln\left(\frac{2r}{d}\right)\right]$$

Show that the no-slip condition is satisfied by this expression. Find the location at which the shear stress is zero, and the stress on the tube and on the filament. Plot the velocity distribution and the stress distribution. (For the stress curve, set an upper limit on stress of 5 Pa.) Discuss the results.

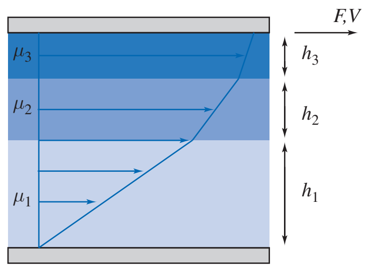
2.56 Fluids of viscosities $\mu_1 = 0.1\ \mathrm{N \cdot s/m^2}$ and $\mu_2 = 0.15\ \mathrm{N \cdot s/m^2}$ are contained between two plates (each plate is 1 m^2 in area).



P2.56

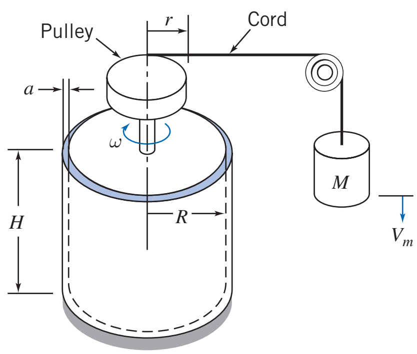
The thicknesses are $h_1 = 0.5$ mm and $h_2 = 0.3$ mm, respectively. Find the force F to make the upper plate move at a speed of 1 m/s. What is the fluid velocity at the interface between the two fluids?

2.57 Fluids of viscosities $\mu_1 = 0.15\ \mathrm{N \cdot s/m^2}$, $\mu_2 = 0.5\ \mathrm{N \cdot s/m^2}$, and $\mu_3 = 0.2\ \mathrm{N \cdot s/m^2}$ are contained between two plates (each plate is 1 m^2 in area). The thicknesses are $h_1 = 0.5$ mm, $h_2 = 0.25$ mm, and $h_3 = 0.2$ mm, respectively. Find the steady speed V of the upper plate and the velocities at the two interfaces due to a force $F = 100$ N. Plot the velocity distribution.



P2.57

2.58 A concentric cylinder viscometer may be formed by rotating the inner member of a pair of closely fitting cylinders. The annular gap is small so that a linear velocity profile will exist in the liquid sample. Consider a viscometer with an inner cylinder of 4 in. diameter and 8 in. height, and a clearance gap width of 0.001 in., filled with castor oil at 90°F. Determine the torque required to turn the inner cylinder at 400 rpm.



P2.58, P2.59, P2.60, P2.61

2.59 A concentric cylinder viscometer may be formed by rotating the inner member of a pair of closely fitting cylinders. For small clearances, a linear velocity profile may be assumed in the liquid filling the annular clearance gap. A viscometer has an inner cylinder of 75 mm diameter and 150 mm height, with a clearance gap width of 0.02 mm. A torque of 0.021 N · m is required to turn the inner cylinder at 100 rpm. Determine the viscosity of the liquid in the clearance gap of the viscometer.

2.60 A concentric cylinder viscometer is driven by a falling mass M connected by a cord and pulley to the inner cylinder, as shown. The liquid to be tested fills the annular gap of width a and height H. After a brief starting transient, the mass falls at constant speed V_m. Develop an algebraic expression for

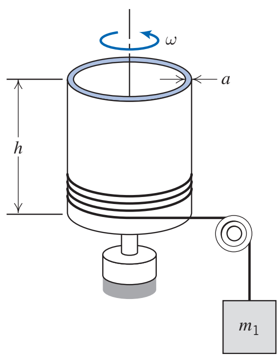
the viscosity of the liquid in the device in terms of M, g, V_m, r, R, a, and H. Evaluate the viscosity of the liquid using:

$$M = 0.10 \text{ kg} \qquad r = 25 \text{ mm}$$
$$R = 50 \text{ mm} \qquad a = 0.20 \text{ mm}$$
$$H = 80 \text{ mm} \qquad V_m = 30 \text{ mm/s}$$

2.61 The viscometer of Problem 2.60 is being used to verify that the viscosity of a particular fluid is $\mu = 0.1\ \text{N}\cdot\text{s/m}^2$. Unfortunately the cord snaps during the experiment. How long will it take the cylinder to lose 99% of its speed? The moment of inertia of the cylinder/pulley system is $0.0273\ \text{kg}\cdot\text{m}^2$.

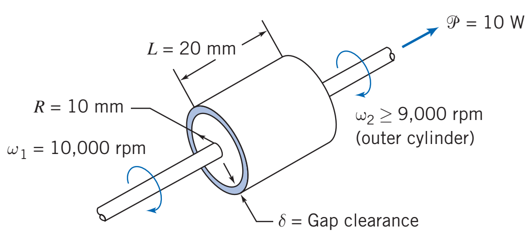
2.62 A shaft with outside diameter of 18 mm turns at 20 revolutions per second inside a stationary journal bearing 60 mm long. A thin film of oil 0.2 mm thick fills the concentric annulus between the shaft and journal. The torque needed to turn the shaft is 0.0036 N·m. Estimate the viscosity of the oil that fills the gap.

2.63 The thin outer cylinder (mass m_2 and radius R) of a small portable concentric cylinder viscometer is driven by a falling mass, m_1, attached to a cord. The inner cylinder is stationary. The clearance between the cylinders is a. Neglect bearing friction, air resistance, and the mass of liquid in the viscometer. Obtain an algebraic expression for the torque due to viscous shear that acts on the cylinder at angular speed ω. Derive and solve a differential equation for the angular speed of the outer cylinder as a function of time. Obtain an expression for the maximum angular speed of the cylinder.



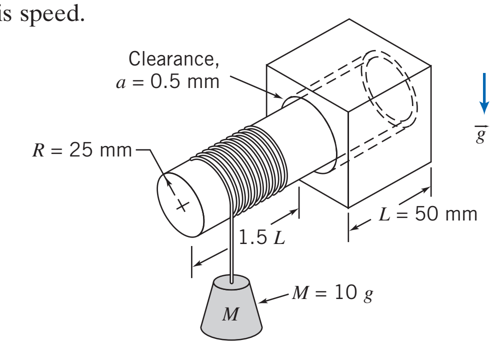
P2.63

2.64 A shock-free coupling for a low-power mechanical drive is to be made from a pair of concentric cylinders. The annular space between the cylinders is to be filled with oil. The drive must transmit power, $\mathcal{P} = 10$ W. Other dimensions and properties are as shown. Neglect any bearing friction and end effects. Assume the minimum practical gap clearance δ for the device is $\delta = 0.25$ mm. Dow manufactures silicone fluids with viscosities as high as 10^6 centipoise. Determine the viscosity that should be specified to satisfy the requirement for this device.



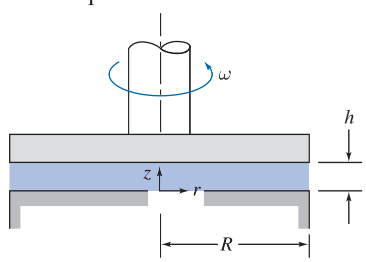
P2.64

2.65 A circular aluminum shaft mounted in a journal is shown. The symmetric clearance gap between the shaft and journal is filled with SAE 10W-30 oil at $T = 30°$C. The shaft is caused to turn by the attached mass and cord. Develop and solve a differential equation for the angular speed of the shaft as a function of time. Calculate the maximum angular speed of the shaft and the time required to reach 95 percent of this speed.



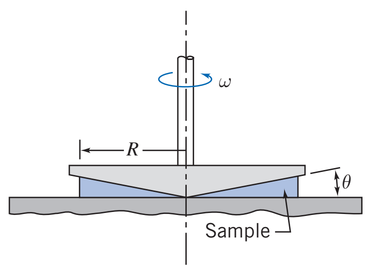
P2.65

2.66 A proposal has been made to use a pair of parallel disks to measure the viscosity of a liquid sample. The upper disk rotates at height h above the lower disk. The viscosity of the liquid in the gap is to be calculated from measurements of the torque needed to turn the upper disk steadily. Obtain an algebraic expression for the torque needed to turn the disk. Could we use this device to measure the viscosity of a non-Newtonian fluid? Explain.



P2.66

2.67 The cone and plate viscometer shown is an instrument used frequently to characterize non-Newtonian fluids. It consists of a flat plate and a rotating cone with a very obtuse angle (typically θ is less than 0.5 degrees). The apex of the cone just touches the plate surface and the liquid to be tested fills the narrow gap formed by the cone and plate. Derive an expression for the shear rate in the liquid that fills the gap in terms of the geometry of the system. Evaluate the torque on the driven cone in terms of the shear stress and geometry of the system.



P2.67, P2.68

2.68 The viscometer of Problem 2.67 is used to measure the apparent viscosity of a fluid. The data below are obtained. What kind of non-Newtonian fluid is this? Find the values of k and n used in Eqs. 2.16 and 2.17 in defining the apparent viscosity of a fluid. (Assume θ is 0.5 degrees.) Predict the viscosity at 90 and 100 rpm.

Speed (rpm)	10	20	30	40	50	60	70	80
μ **(N·s/m^2)**	0.121	0.139	0.153	0.159	0.172	0.172	0.183	0.185

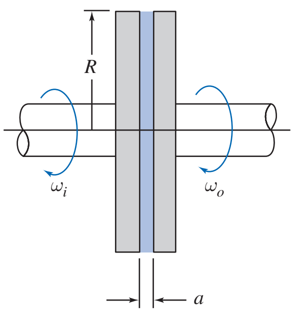
2.69 An insulation company is examining a new material for extruding into cavities. The experimental data is given below for the speed U of the upper plate, which is separated from a fixed lower plate by a 1-mm-thick sample of the material, when a given shear stress is applied. Determine the type of material. If a replacement material with a minimum yield stress of 250 Pa is needed, what viscosity will the material need to have the same behavior as the current material at a shear stress of 450 Pa?

τ **(Pa)**	50	100	150	163	171	170	202	246	349	444
U **(m/s)**	0	0	0	0.005	0.01	0.025	0.05	0.1	0.2	0.3

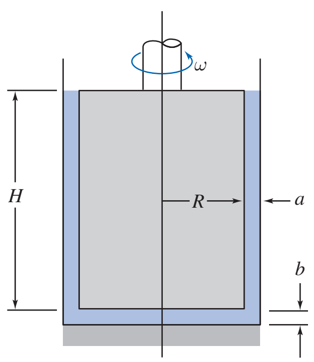
2.70 A viscometer is used to measure the viscosity of a patient's blood. The deformation rate (shear rate)–shear stress data is shown below. Plot the apparent viscosity versus deformation rate. Find the value of k and n in Eq. 2.17, and from this examine the aphorism "Blood is thicker than water."

du/dy **(s^{-1})**	5	10	25	50	100	200	300	400
τ **(Pa)**	0.0457	0.119	0.241	0.375	0.634	1.06	1.46	1.78

2.71 A viscous clutch is to be made from a pair of closely spaced parallel disks enclosing a thin layer of viscous liquid. Develop algebraic expressions for the torque and the power transmitted by the disk pair, in terms of liquid viscosity, μ, disk radius, R, disk spacing, a, and the angular speeds: ω_i of the input disk and ω_o of the output disk. Also develop expressions for the slip ratio, $s = \Delta\omega/\omega_i$, in terms of ω_i and the torque transmitted. Determine the efficiency, η, in terms of the slip ratio.



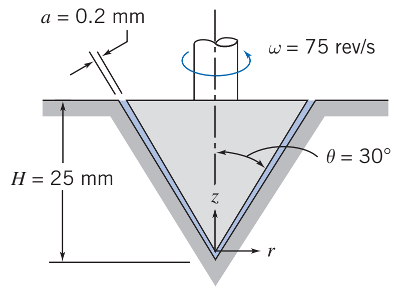
P2.71



P2.72

2.72 A concentric-cylinder viscometer is shown. Viscous torque is produced by the annular gap around the inner cylinder. Additional viscous torque is produced by the flat bottom of the inner cylinder as it rotates above the flat bottom of the stationary outer cylinder. Obtain an algebraic expression for the viscous torque due to flow in the annular gap of width a. Obtain an algebraic expression for the viscous torque due to flow in the bottom clearance gap of height b. Prepare a plot showing the ratio, b/a, required to hold the bottom torque to 1 percent or less of the annulus torque, versus the other geometric variables. What are the design implications? What modifications to the design can you recommend?

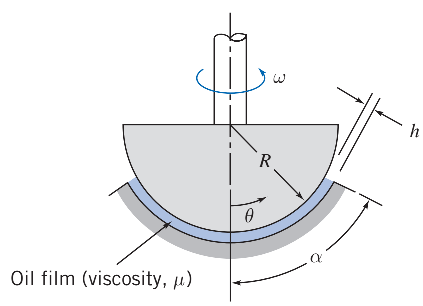
2.73 A viscometer is built from a conical pointed shaft that turns in a conical bearing, as shown. The gap between shaft and bearing is filled with a sample of the test oil. Obtain an algebraic expression for the viscosity μ of the oil as a function of viscometer geometry (H, a, and θ), turning speed ω, and applied torque T. For the data given, find by referring to Figure A.2 in Appendix A, the type of oil for which the applied torque is 0.325 N·m. The oil is at 20°C. *Hint*: First obtain an expression for the shear stress on the surface of the conical shaft as a function of z.



P2.73

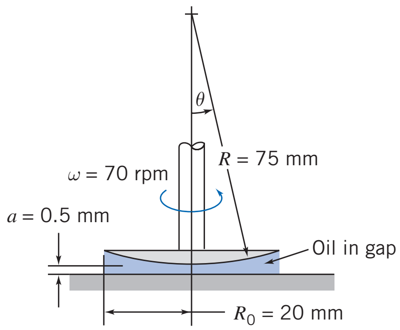
2.74 Design a concentric-cylinder viscometer to measure the viscosity of a liquid similar to water. The goal is to achieve a measurement accuracy of ±1 percent. Specify the configuration and dimensions of the viscometer. Indicate what measured parameter will be used to infer the viscosity of the liquid sample.

2.75 A spherical thrust bearing is shown. The gap between the spherical member and the housing is of constant width h. Obtain and plot an algebraic expression for the nondimensional torque on the spherical member, as a function of angle α.



P2.75

2.76 A cross section of a rotating bearing is shown. The spherical member rotates with angular speed ω, a small distance, a, above the plane surface. The narrow gap is filled with viscous oil, having $\mu = 1250$ cp. Obtain an algebraic expression for the shear stress acting on the spherical member. Evaluate the maximum shear stress that acts on the spherical member for the conditions shown. (Is the maximum necessarily located at the maximum radius?) Develop an algebraic expression (in the form of an integral) for the total viscous shear torque that acts on the spherical member. Calculate the torque using the dimensions shown.



P 2.76

Surface Tension

2.77 Small gas bubbles form in soda when a bottle or can is opened. The average bubble diameter is about 0.1 mm. Estimate the pressure difference between the inside and outside of such a bubble.

2.78 You intend to gently place several steel needles on the free surface of the water in a large tank. The needles come in two lengths: Some are 5 cm long, and some are 10 cm long. Needles of each length are available with diameters of 1 mm, 2.5 mm, and 5 mm. Make a prediction as to which needles, if any, will float.

2.79 According to Folsom [6], the capillary rise Δh (in.) of a water-air interface in a tube is correlated by the following empirical expression:

$$\Delta h = Ae^{-b \cdot D}$$

where D (in.) is the tube diameter, $A = 0.400$, and $b = 4.37$. You do an experiment to measure Δh versus D and obtain:

D (in.)	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8	0.9	1	1.1
Δh (in.)	0.232	0.183	0.09	0.059	0.052	0.033	0.017	0.01	0.006	0.004	0.003

What are the values of A and b that best fit this data using *Excel*'s *Trendline* feature? Do they agree with Folsom's values? How good is the data?

2.80 Slowly fill a glass with water to the maximum possible level. Observe the water level closely. Explain how it can be higher than the rim of the glass.

2.81 Plan an experiment to measure the surface tension of a liquid similar to water. If necessary, review the NCFMF video *Surface Tension* for ideas. Which method would be most suitable for use in an undergraduate laboratory? What experimental precision could be expected?

Description and Classification of Fluid Motions

2.82 Water usually is assumed to be incompressible when evaluating static pressure variations. Actually it is 100 times more compressible than steel. Assuming the bulk modulus of water is constant, compute the percentage change in density for water raised to a gage pressure of 100 atm. Plot the percentage change in water density as a function of p/p_{atm} up to a pressure of 50,000 psi, which is the approximate pressure used for high-speed cutting jets of water to cut concrete and other composite materials. Would constant density be a reasonable assumption for engineering calculations for cutting jets?

2.83 The viscous boundary layer velocity profile shown in Fig. 2.15 can be approximated by a parabolic equation,

$$u(y) = a + b\left(\frac{y}{\delta}\right) + c\left(\frac{y}{\delta}\right)^2$$

The boundary condition is $u = U$ (the free stream velocity) at the boundary edge δ (where the viscous friction becomes zero). Find the values of a, b, and c.

2.84 The viscous boundary layer velocity profile shown in Fig. 2.15 can be approximated by a cubic equation,

$$u(y) = a + b\left(\frac{y}{\delta}\right) + c\left(\frac{y}{\delta}\right)^3$$

The boundary condition is $u = U$ (the free stream velocity) at the boundary edge δ (where the viscous friction becomes zero). Find the values of a, b, and c.

2.85 At what minimum speed (in mph) would an automobile have to travel for compressibility effects to be important? Assume the local air temperature is 60°F.

2.86 In a food industry process, carbon tetrachloride at 20°C flows through a tapered nozzle from an inlet diameter D_{in} = 50 mm to an outlet diameter of D_{out}. The area varies linearly with distance along the nozzle, and the exit area is one-fifth of the inlet area; the nozzle length is 250 mm. The flow rate is $Q = 2$ L/min. It is important for the process that the flow exits the nozzle as a turbulent flow. Does it? If so, at what point along the nozzle does the flow become turbulent?

2.87 What is the Reynolds number of water at 20°C flowing at 0.25 m/s through a 5-mm-diameter tube? If the pipe is now heated, at what mean water temperature will the flow transition to turbulence? Assume the velocity of the flow remains constant.

2.88 A supersonic aircraft travels at 2700 km/hr at an altitude of 27 km. What is the Mach number of the aircraft? At what approximate distance measured from the leading edge of the aircraft's wing does the boundary layer change from laminar to turbulent?

2.89 SAE 30 oil at 100°C flows through a 12-mm-diameter stainless-steel tube. What is the specific gravity and specific

weight of the oil? If the oil discharged from the tube fills a 100-mL graduated cylinder in 9 seconds, is the flow laminar or turbulent?

2.90 A seaplane is flying at 100 mph through air at 45°F. At what distance from the leading edge of the underside of the fuselage does the boundary layer transition to turbulence? How does this boundary layer transition change as the underside of the fuselage touches the water during landing? Assume the water temperature is also 45°F.

2.91 An airliner is cruising at an altitude of 5.5 km with a speed of 700 km/hr. As the airliner increases its altitude, it adjusts its speed so that the Mach number remains constant. Provide a sketch of speed vs. altitude. What is the speed of the airliner at an altitude of 8 km?

2.92 How does an airplane wing develop lift?

3

Fluid Statics

Case Study in Energy and the Environment

Wave Power: *Wavebob*

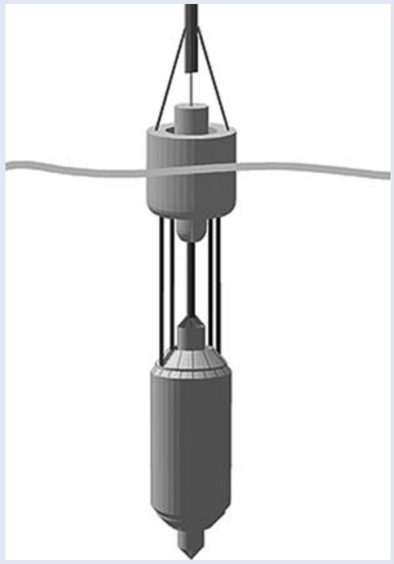
Humans have been interested in tapping the immense power of the ocean for centuries, but with fossil fuels (oil and gas) becoming depleted, the development of ocean energy technology is becoming important. Wave power in particular is attractive to a number of countries with access to a suitable resource. Geographically and commercially it's believed the richest wave energy resources currently are off the Atlantic coast of Europe (in particular near Ireland, the UK, and Portugal), the west coast of North America (from San Francisco to British Columbia), Hawaii, and New Zealand.

A family of devices called *point absorbers* is being developed by a number of companies. These are usually axisymmetric about a vertical axis, and by definition they are small compared to the wavelength of the waves that they are designed to exploit. The devices usually operate in a vertical mode, often referred to as *heave*; a surface-piercing float rises and falls with the passing waves and reacts against either the seabed or something attached to it. These devices ultimately depend on a buoyancy force, one of the topics of this chapter.

A company named *Wavebob Ltd*. has developed one of the simplest of these devices. This innovative

eponymous device, as shown in the figure, is proving to be successful for extracting wave energy. The figure does not indicate the size of the device, but it is quite large; the upper champer has a diameter of 20 m. It looks like just another buoy floating on the surface, but underneath it is constantly harvesting energy. The lower component of the *Wavebob* is tethered to the ocean floor and so remains in its vertical location, while the section at the surface oscillates as the waves move over it. Hence the distance between the two components is constantly changing, with a significant force between them; work can thus be done on an electrical generator. The two components of the machinery contain electronic systems that can be controlled remotely or self-regulating, and these make the internal mechanism automatically react to changing ocean and wave conditions by retuning as needed, so that at all times the maximum amount of energy is harvested.

It has already been tested in the Atlantic Ocean off the coast of Ireland and is designed to have a 25-year life span and to be able to survive all but the very worst storms. Each *Wavebob* is expected to produce about 500 kW of power or more, sufficient electricity for over a thousand homes; it is intended to be part of a large array of such devices. It seems likely this device will become ubiquitous because it is relatively inexpensive, very low maintenance, and durable, and it takes up only a small area.



Schematic of *Wavebob* (Picture courtesy of Gráinne Byrne, *Wavebob* Ltd.)

In Chapter 1, we defined a fluid as any substance that flows (continuously deforms) when it experiences a shear stress; hence for a static fluid (or one undergoing "rigid-body" motion) only normal stress is present—in other words, pressure. We will study the topic of fluid statics (often called *hydrostatics*, even though it is not restricted to water) in this chapter.

Although fluid statics problems are the simplest kind of fluid mechanics problems, this is not the only reason we will study them. The pressure generated within a static fluid is an important phenomenon in many practical situations. Using the principles of hydrostatics, we can compute forces on submerged objects, develop instruments for measuring pressures, and deduce properties of the atmosphere and oceans. The principles of hydrostatics also may be used to determine the forces developed by hydraulic systems in applications such as industrial presses or automobile brakes.

In a static, homogeneous fluid, or in a fluid undergoing rigid-body motion, a fluid particle retains its identity for all time, and fluid elements do not deform. We may apply Newton's second law of motion to evaluate the forces acting on the particle.

3.1 The Basic Equation of Fluid Statics

The first objective of this chapter is to obtain an equation for computing the pressure field in a static fluid. We will deduce what we already know from everyday experience, that the pressure increases with depth. To do this, we apply Newton's second law to a differential fluid element of mass $dm = \rho\, d\forall$, with sides dx, dy, and dz, as shown

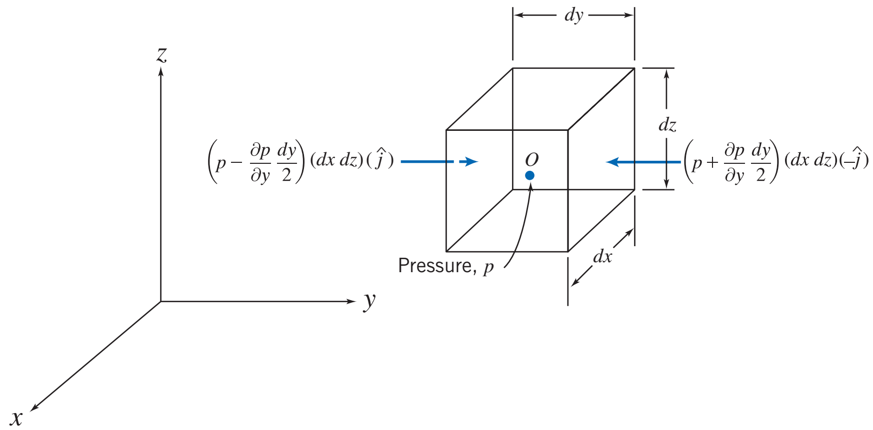


Fig. 3.1 Differential fluid element and pressure forces in the y direction.

in Fig. 3.1. The fluid element is stationary relative to the stationary rectangular coordinate system shown. (Fluids in rigid-body motion will be treated in Section 3.7 on the Web.)

From our previous discussion, recall that two general types of forces may be applied to a fluid: body forces and surface forces. The only body force that must be considered in most engineering problems is due to gravity. In some situations body forces caused by electric or magnetic fields might be present; they will not be considered in this text.

CLASSIC VIDEO

Magnetohydrodynamics.

For a differential fluid element, the body force is

$$d\vec{F}_B = \vec{g}\,dm = \vec{g}\rho\,d\forall$$

where $\vec{g}$ is the local gravity vector, ρ is the density, and $d\forall$ is the volume of the element. In Cartesian coordinates $d\forall = dx\,dy\,dz$, so

$$d\vec{F}_B = \rho\vec{g}\,dx\,dy\,dz$$

In a static fluid there are no shear stresses, so the only surface force is the pressure force. Pressure is a scalar field, $p = p(x, y, z)$; in general we expect the pressure to vary with position within the fluid. The net pressure force that results from this variation can be found by summing the forces that act on the six faces of the fluid element.

Let the pressure be p at the center, O, of the element. To determine the pressure at each of the six faces of the element, we use a Taylor series expansion of the pressure about point O. The pressure at the left face of the differential element is

$$p_L = p + \frac{\partial p}{\partial y}(y_L - y) = p + \frac{\partial p}{\partial y}\left(-\frac{dy}{2}\right) = p - \frac{\partial p}{\partial y}\frac{dy}{2}$$

(Terms of higher order are omitted because they will vanish in the subsequent limiting process.) The pressure on the right face of the differential element is

$$p_R = p + \frac{\partial p}{\partial y}(y_R - y) = p + \frac{\partial p}{\partial y}\frac{dy}{2}$$

The pressure *forces* acting on the two y surfaces of the differential element are shown in Fig. 3.1. Each pressure force is a product of three factors. The first is the magnitude of the pressure. This magnitude is multiplied by the area of the face to give the magnitude of the pressure force, and a unit vector is introduced to indicate direction. Note also in Fig. 3.1 that the pressure force on each face acts *against* the face. A positive pressure corresponds to a *compressive* normal stress.

Pressure forces on the other faces of the element are obtained in the same way. Combining all such forces gives the net surface force acting on the element. Thus

$$d\vec{F}_S = \left(p - \frac{\partial p}{\partial x}\frac{dx}{2}\right)(dy\,dz)(\hat{i}) + \left(p + \frac{\partial p}{\partial x}\frac{dx}{2}\right)(dy\,dz)(-\hat{i})$$
$$+ \left(p - \frac{\partial p}{\partial y}\frac{dy}{2}\right)(dx\,dz)(\hat{j}) + \left(p + \frac{\partial p}{\partial y}\frac{dy}{2}\right)(dx\,dz)(-\hat{j})$$
$$+ \left(p - \frac{\partial p}{\partial z}\frac{dz}{2}\right)(dx\,dy)(\hat{k}) + \left(p + \frac{\partial p}{\partial z}\frac{dz}{2}\right)(dx\,dy)(-\hat{k})$$

Collecting and canceling terms, we obtain

$$d\vec{F}_S = -\left(\frac{\partial p}{\partial x}\hat{i} + \frac{\partial p}{\partial y}\hat{j} + \frac{\partial p}{\partial z}\hat{k}\right)dx\,dy\,dz \tag{3.1a}$$

The term in parentheses is called the gradient of the pressure or simply the pressure gradient and may be written grad p or ∇p. In rectangular coordinates

$$\text{grad}\,p \equiv \nabla p \equiv \left(\hat{i}\frac{\partial p}{\partial x} + \hat{j}\frac{\partial p}{\partial y} + \hat{k}\frac{\partial p}{\partial z}\right) \equiv \left(\hat{i}\frac{\partial}{\partial x} + \hat{j}\frac{\partial}{\partial y} + \hat{k}\frac{\partial}{\partial z}\right)p$$

The gradient can be viewed as a vector operator; taking the gradient of a scalar field gives a vector field. Using the gradient designation, Eq. 3.1a can be written as

$$d\vec{F}_S = -\text{grad}\,p\,(dx\,dy\,dz) = -\nabla p\,dx\,dy\,dz \tag{3.1b}$$

Physically the gradient of pressure is the negative of the surface force per unit volume due to pressure. Note that the pressure magnitude itself is not relevant in computing the net pressure force; instead what counts is the rate of change of pressure with distance, the *pressure gradient*. We shall encounter this term throughout our study of fluid mechanics.

We combine the formulations for surface and body forces that we have developed to obtain the total force acting on a fluid element. Thus

$$d\vec{F} = d\vec{F}_S + d\vec{F}_B = (-\nabla p + \rho\vec{g})\,dx\,dy\,\mathrm{dz} = (-\nabla p + \rho\vec{g})\,d\forall$$

or on a per unit volume basis

$$\frac{d\vec{F}}{d\forall} = -\nabla p + \rho\vec{g} \tag{3.2}$$

For a fluid particle, Newton's second law gives $\vec{F} = \vec{a}\,dm = \vec{a}\rho d\forall$. For a static fluid, $\vec{a} = 0$. Thus

$$\frac{d\vec{F}}{d\forall} = \rho\vec{a} = 0$$

Substituting for $d\vec{F}/d\forall$ from Eq. 3.2, we obtain

$$-\nabla p + \rho\vec{g} = 0 \tag{3.3}$$

Let us review this equation briefly. The physical significance of each term is

$$\begin{array}{ccccc} -\nabla p & + & \rho\vec{g} & = 0 \\ \left\{\begin{array}{c}\text{net pressure force}\\ \text{per unit volume}\\ \text{at a point}\end{array}\right\} & + & \left\{\begin{array}{c}\text{body force per}\\ \text{unit volume}\\ \text{at a point}\end{array}\right\} & = 0 \end{array}$$

This is a vector equation, which means that it is equivalent to three component equations that must be satisfied individually. The component equations are

$$\left.\begin{aligned} -\frac{\partial p}{\partial x} + \rho g_x &= 0 \quad x \text{ direction} \\ -\frac{\partial p}{\partial y} + \rho g_y &= 0 \quad y \text{ direction} \\ -\frac{\partial p}{\partial z} + \rho g_z &= 0 \quad z \text{ direction} \end{aligned}\right\} \tag{3.4}$$

Equations 3.4 describe the pressure variation in each of the three coordinate directions in a static fluid. It is convenient to choose a coordinate system such that the gravity vector is aligned with one of the coordinate axes. If the coordinate system is chosen with the z axis directed vertically upward, as in Fig. 3.1, then $g_x = 0$, $g_y = 0$, and $g_z = -g$. Under these conditions, the component equations become

$$\frac{\partial p}{\partial x} = 0 \qquad \frac{\partial p}{\partial y} = 0 \qquad \frac{\partial p}{\partial z} = -\rho g \tag{3.5}$$

Equations 3.5 indicate that, under the assumptions made, the pressure is independent of coordinates x and y; it depends on z alone. Thus since p is a function of a single variable, a total derivative may be used instead of a partial derivative. With these simplifications, Eqs. 3.5 finally reduce to

$$\frac{dp}{dz} = -\rho g \equiv -\gamma \tag{3.6}$$

Restrictions: (1) Static fluid.
(2) Gravity is the only body force.
(3) The z axis is vertical and upward.

In Eq. 3.6, γ is the specific weight of the fluid. This equation is the basic pressure-height relation of fluid statics. It is subject to the restrictions noted. Therefore it must be applied only where these restrictions are reasonable for the physical situation. To determine the pressure distribution in a static fluid, Eq. 3.6 may be integrated and appropriate boundary conditions applied.

Before considering specific applications of this equation, it is important to remember that pressure values must be stated with respect to a reference level. If the reference level is a vacuum, pressures are termed *absolute*, as shown in Fig. 3.2.

Most pressure gages indicate a pressure *difference*—the difference between the measured pressure and the ambient level (usually atmospheric pressure). Pressure levels measured with respect to atmospheric pressure are termed *gage* pressures. Thus

$$p_{\text{gage}} = p_{\text{absolute}} - p_{\text{atmosphere}}$$

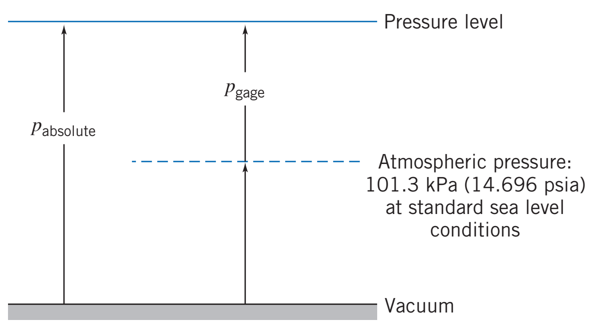


Fig. 3.2 Absolute and gage pressures, showing reference levels.

For example, a tire gage might indicate 30 psi; the absolute pressure would be about 44.7 psi. Absolute pressures must be used in all calculations with the ideal gas equation or other equations of state.

3.2 The Standard Atmosphere

Scientists and engineers sometimes need a numerical or analytical model of the Earth's atmosphere in order to simulate climate variations to study, for example, effects of global warming. There is no single standard model. An International Standard Atmosphere (ISA) has been defined by the International Civil Aviation Organization (ICAO); there is also a similar U.S. Standard Atmosphere.

The temperature profile of the U.S. Standard Atmosphere is shown in Fig. 3.3. Additional property values are tabulated as functions of elevation in Appendix A. Sea level conditions of the U.S. Standard Atmosphere are summarized in Table 3.1.

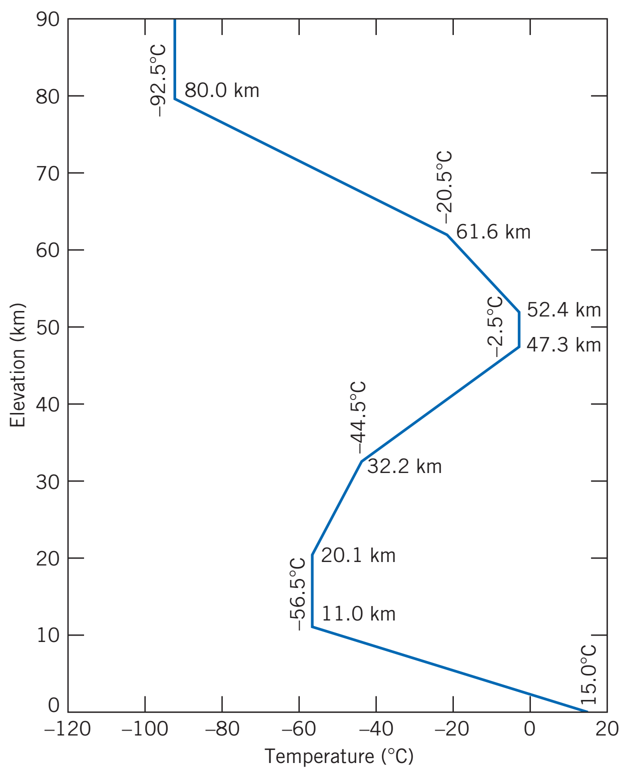


Fig. 3.3 Temperature variation with altitude in the U.S. Standard Atmosphere.

Table 3.1
Sea Level Conditions of the U.S. Standard Atmosphere

Property	Symbol	SI	English
Temperature	T	15°C	59°F
Pressure	p	101.3 kPa (abs)	14.696 psia
Density	ρ	1.225 kg/m^3	0.002377 slug/ft^3
Specific weight	γ	—	0.07651 lbf/ft^3
Viscosity	μ	1.789×10^{-5} kg/(m·s) (Pa·s)	3.737×10^{-7} lbf·s/ft^2

Pressure Variation in a Static Fluid 3.3

We proved that pressure variation in any static fluid is described by the basic pressure-height relation

$$\frac{dp}{dz} = -\rho g \tag{3.6}$$

Although ρg may be defined as the specific weight, γ, it has been written as ρg in Eq. 3.6 to emphasize that *both* ρ and g must be considered variables. In order to integrate Eq. 3.6 to find the pressure distribution, we need information about variations in both ρ and g.

For most practical engineering situations, the variation in g is negligible. Only for a purpose such as computing very precisely the pressure change over a large elevation difference would the variation in g need to be included. Unless we state otherwise, we shall assume g to be constant with elevation at any given location.

Incompressible Liquids: Manometers

For an incompressible fluid, $\rho = \text{constant}$. Then for constant gravity,

$$\frac{dp}{dz} = -\rho g = \text{constant}$$

To determine the pressure variation, we must integrate and apply appropriate boundary conditions. If the pressure at the reference level, z_0, is designated as p_0, then the pressure, p, at level z is found by integration:

$$\int_{p_0}^{p} dp = -\int_{z_0}^{z} \rho g \, dz$$

or

$$p - p_0 = -\rho g(z - z_0) = \rho g(z_0 - z)$$

For liquids, it is often convenient to take the origin of the coordinate system at the free surface (reference level) and to measure distances as positive downward from the free surface as in Fig. 3.4.

With h measured positive downward, we have

$$z_0 - z = h$$

and obtain

$$p - p_0 = \Delta p = \rho g h \tag{3.7}$$

Equation 3.7 indicates that the pressure difference between two points in a static incompressible fluid can be determined by measuring the elevation difference between the two points. Devices used for this purpose are called *manometers*.

Use of Eq. 3.7 for a manometer is illustrated in Example 3.1.

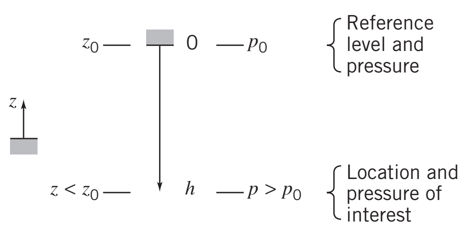
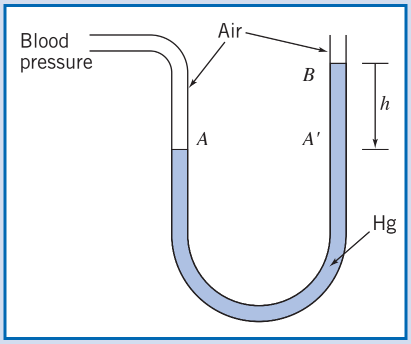


Fig. 3.4 Use of z and h coordinates.

Example 3.1 SYSTOLIC AND DIASTOLIC PRESSURE

Normal blood pressure for a human is 120/80 mm Hg. By modeling a sphygmomanometer pressure gage as a U-tube manometer, convert these pressures to psig.



Given: Gage pressures of 120 and 80 mm Hg.

Find: The corresponding pressures in psig.

Solution:
Apply hydrostatic equation to points A, A', and B.

Governing equation:

$$p - p_0 = \Delta p = \rho g h \tag{3.7}$$

Assumptions: (1) Static fluid.
(2) Incompressible fluids.
(3) Neglect air density ($\ll$ Hg density).

Applying the governing equation between points A' and B (and p_{B} is atmospheric and therefore zero gage):

$$p_{A'} = p_B + \rho_{\mathrm{Hg}} g h = SG_{\mathrm{Hg}} \rho_{\mathrm{H_2O}} g h$$

In addition, the pressure increases as we go downward from point A' to the bottom of the manometer, and decreases by an equal amount as we return up the left branch to point A. This means points A and A' have the same pressure, so we end up with

$$p_A = p_{A'} = SG_{\mathrm{Hg}} \rho_{\mathrm{H_2O}} g h$$

Substituting $SG_{\mathrm{Hg}} = 13.6$ and $\rho_{\mathrm{H_2O}} = 1.94$ slug/ft^3 from Appendix A.1 yields for the systolic pressure ($h = 120$ mm Hg)

$$p_{\text{systolic}} = p_A = 13.6 \times 1.94 \frac{\text{slug}}{\text{ft}^3} \times 32.2 \frac{\text{ft}}{\text{s}^2} \times 120\,\text{mm} \times \frac{\text{in.}}{25.4\,\text{mm}} \times \frac{\text{ft}}{12\,\text{in.}} \times \frac{\text{lbf}\cdot\text{s}^2}{\text{slug}\cdot\text{ft}}$$

$$p_{\text{systolic}} = 334\,\text{lbf/ft}^2 = 2.32\ \text{psi} \longleftarrow p_{\text{systolic}}$$

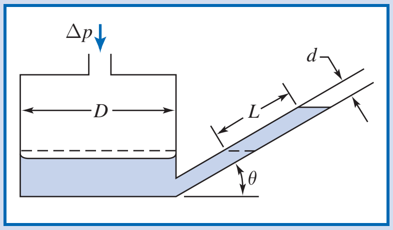
By a similar process, the diastolic pressure ($h = 80$ mm Hg) is

$$p_{\text{diastolic}} = 1.55\,\text{psi} \longleftarrow p_{\text{diastolic}}$$

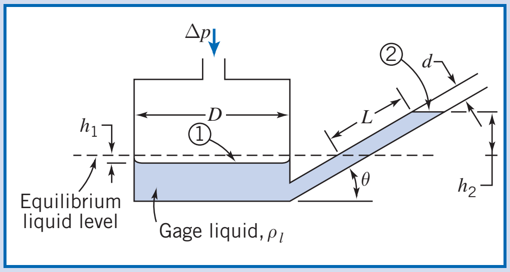
Notes:
- ✓ Two points at the same level in a continuous single fluid have the same pressure.
- ✓ In manometer problems we neglect change in pressure with depth for a gas: $\rho_{\text{gas}} \ll \rho_{\text{liquid}}$.
- ✓ This problem shows the conversion from mm Hg to psi, using Eq. 3.7: 120 mm Hg is equivalent to about 2.32 psi. More generally, 1 atm = 14.7 psi = 101 kPa = 760 mm Hg.

Manometers are simple and inexpensive devices used frequently for pressure measurements. Because the liquid level change is small at low pressure differential, a U-tube manometer may be difficult to read accurately. The *sensitivity* of a manometer is a measure of how sensitive it is compared to a simple water-filled U-tube manometer. Specifically, it is the ratio of the deflection of the manometer to that of a water-filled U-tube manometer, due to the same applied pressure difference Δp. Sensitivity can be increased by changing the manometer design or by using two immiscible liquids of slightly different density. Analysis of an inclined manometer is illustrated in Example 3.2.

Example 3.2 ANALYSIS OF INCLINED-TUBE MANOMETER



An inclined-tube reservoir manometer is constructed as shown. Derive a general expression for the liquid deflection, L, in the inclined tube, due to the applied pressure difference, Δp. Also obtain an expression for the manometer sensitivity, and discuss the effect on sensitivity of D, d, θ, and SG.



Given: Inclined-tube reservoir manometer.

Find: Expression for L in terms of Δp.
General expression for manometer sensitivity.
Effect of parameter values on sensitivity.

Solution:
Use the equilibrium liquid level as a reference.

Governing equations: $p - p_0 = \Delta p = \rho g h \qquad SG = \dfrac{\rho}{\rho_{H_2O}}$

Assumptions: (1) Static fluid.
(2) Incompressible fluid.

Applying the governing equation between points *1* and *2*

$$p_1 - p_2 = \Delta p = \rho_l g(h_1 + h_2) \tag{1}$$

To eliminate h_1, we recognize that the *volume* of manometer liquid remains constant; the volume displaced from the reservoir must equal the volume that rises in the tube, so

$$\frac{\pi D^2}{4} h_1 = \frac{\pi d^2}{4} L \quad \text{or} \quad h_1 = L\left(\frac{d}{D}\right)^2$$

In addition, from the geometry of the manometer, $h_2 = L \sin \theta$. Substituting into Eq. 1 gives

$$\Delta p = \rho_l g\left[L \sin \theta + L\left(\frac{d}{D}\right)^2\right] = \rho_l g L\left[\sin \theta + \left(\frac{d}{D}\right)^2\right]$$

Thus

$$L = \frac{\Delta p}{\rho_l g\left[\sin \theta + \left(\dfrac{d}{D}\right)^2\right]} \quad \longleftarrow L$$

To find the sensitivity of the manometer, we need to compare this to the deflection h a simple U-tube manometer, using water (density ρ), would experience,

$$h = \frac{\Delta p}{\rho g}$$

The sensitivity s is then

$$s = \frac{L}{h} = \frac{1}{SG_l\left[\sin \theta + \left(\dfrac{d}{D}\right)^2\right]} \quad \longleftarrow s$$

where we have used $SG_l = \rho_l/\rho$. This result shows that to increase sensitivity, SG_l, $\sin \theta$, and d/D each should be made as small as possible. Thus the designer must choose a gage liquid and two geometric parameters to complete a design, as discussed below.

Gage Liquid

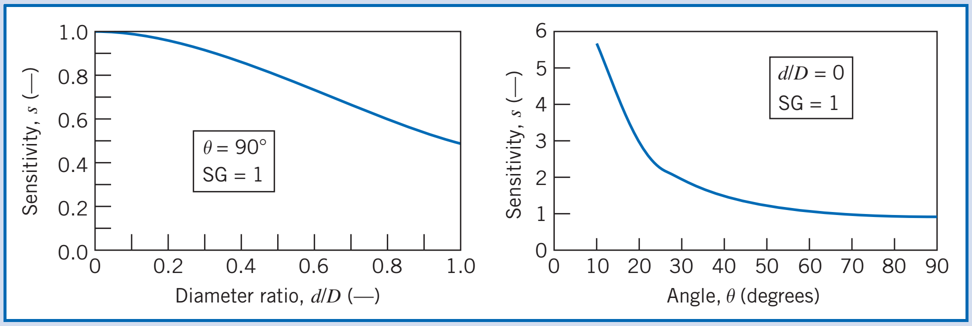
The gage liquid should have the smallest possible specific gravity to increase sensitivity. In addition, the gage liquid must be safe (without toxic fumes or flammability), be immiscible with the fluid being gaged, suffer minimal loss from evaporation, and develop a satisfactory meniscus. Thus the gage liquid should have relatively low surface tension and should accept dye to improve its visibility.

Tables A.1, A.2, and A.4 show that hydrocarbon liquids satisfy many of these criteria. The lowest specific gravity is about 0.8, which increases manometer sensitivity by 25 percent compared to water.

Diameter Ratio

The plot shows the effect of diameter ratio on sensitivity for a vertical reservoir manometer with gage liquid of unity specific gravity. Note that $d/D = 1$ corresponds to an ordinary U-tube manometer; its sensitivity is 0.5 because for this case the total deflection will be h, and for each side it will be $h/2$, so $L = h/2$. Sensitivity doubles to 1.0 as d/D approaches zero because most of the level change occurs in the measuring tube.

The minimum tube diameter d must be larger than about 6 mm to avoid excessive capillary effect. The maximum reservoir diameter D is limited by the size of the manometer. If D is set at 60 mm, so that d/D is 0.1, then



$(d/D)^2 = 0.01$, and the sensitivity increases to 0.99, very close to the maximum attainable value of 1.0.

Inclination Angle

The final plot shows the effect of inclination angle on sensitivity for $d/D = 0$. Sensitivity increases sharply as inclination angle is reduced below 30 degrees. A practical limit is reached at about 10 degrees: The meniscus becomes indistinct and the level hard to read for smaller angles.

Summary

Combining the best values ($SG = 0.8$, $d/D = 0.1$, and $\theta = 10$ degrees) gives a manometer sensitivity of 6.81. Physically this is the ratio of observed gage liquid deflection to equivalent water column height. Thus the deflection in the inclined tube is amplified 6.81 times compared to a vertical water column. With improved sensitivity, a small pressure difference can be read more accurately than with a water manometer, or a smaller pressure difference can be read with the same accuracy.

The graphs were generated from the *Excel* workbook for this Example. This workbook has more detailed graphs, showing sensitivity curves for a range of values of d/D and θ.

Students sometimes have trouble analyzing multiple-liquid manometer situations. The following rules of thumb are useful:

1. Any two points at the same elevation in a continuous region of the same liquid are at the same pressure.
2. Pressure increases as one goes *down* a liquid column (remember the pressure change on diving into a swimming pool).

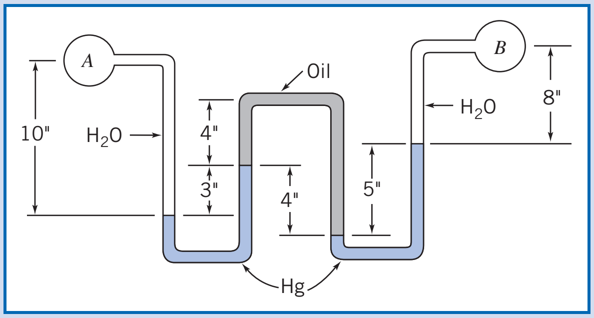
To find the pressure difference Δp between two points separated by a series of fluids, we can use the following modification of Eq. 3.7:

$$\Delta p = g\sum_i \rho_i h_i \tag{3.8}$$

where ρ_i and h_i represent the densities and depths of the various fluids, respectively. Use care in applying signs to the depths h_i; they will be positive downwards, and negative upwards. Example 3.3 illustrates the use of a multiple-liquid manometer for measuring a pressure difference.

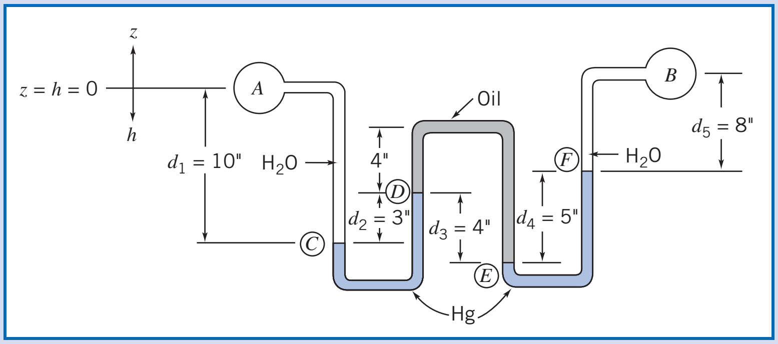
Example 3.3 MULTIPLE-LIQUID MANOMETER

Water flows through pipes A and B. Lubricating oil is in the upper portion of the inverted U. Mercury is in the bottom of the manometer bends. Determine the pressure difference, $p_A - p_B$, in units of lbf/in.^{2}



Given: Multiple-liquid manometer as shown.

Find: Pressure difference, $p_A - p_B$, in lbf/in.^{2}



Solution:

Governing equations: $\Delta p = g\sum_i \rho_i h_i \qquad SG = \dfrac{\rho}{\rho_{H_2O}}$

Assumptions:
(1) Static fluid.
(2) Incompressible fluid.

Applying the governing equation, working from point B to A

$$p_A - p_B = \Delta p = g(\rho_{H_2O}d_5 + \rho_{Hg}d_4 - \rho_{oil}d_3 + \rho_{Hg}d_2 - \rho_{H_2O}d_1) \tag{1}$$

This equation can also be derived by repeatedly using Eq. 3.7 in the following form:

$$p_2 - p_1 = \rho g(h_2 - h_1)$$

Beginning at point A and applying the equation between successive points along the manometer gives

$$\begin{aligned} p_C - p_A &= +\rho_{H_2O} g d_1 \\ p_D - p_C &= -\rho_{Hg} g d_2 \\ p_E - p_D &= +\rho_{oil} g d_3 \\ p_F - p_E &= -\rho_{Hg} g d_4 \\ p_B - p_F &= -\rho_{H_2O} g d_5 \end{aligned}$$

Multiplying each equation by minus one and adding, we obtain Eq. (1)

$$\begin{aligned} p_A - p_B &= (p_A - p_C) + (p_C - p_D) + (p_D - p_E) + (p_E - p_F) + (p_F - p_B) \\ &= -\rho_{H_2O} g d_1 + \rho_{Hg} g d_2 - \rho_{oil} g d_3 + \rho_{Hg} g d_4 + \rho_{H_2O} g d_5 \end{aligned}$$

Substituting $\rho = SG\rho_{H_2O}$ with $SG_{Hg} = 13.6$ and $SG_{oil} = 0.88$ (Table A.2), yields

$$\begin{aligned} p_A - p_B &= g(-\rho_{H_2O} d_1 + 13.6\rho_{H_2O} d_2 - 0.88\rho_{H_2O} d_3 + 13.6\rho_{H_2O} d_4 + \rho_{H_2O} d_5) \\ &= g\rho_{H_2O}(-d_1 + 13.6 d_2 - 0.88 d_3 + 13.6 d_4 + d_5) \end{aligned}$$

$$p_A - p_B = g\rho_{H_2O}(-10 + 40.8 - 3.52 + 68 + 8)\text{ in.}$$

$$p_A - p_B = g\rho_{H_2O} \times 103.3\text{ in.}$$

$$= 32.2\,\frac{\text{ft}}{\text{s}^2} \times 1.94\,\frac{\text{slug}}{\text{ft}^3} \times 103.3\text{ in.} \times \frac{\text{ft}}{12\text{ in.}} \times \frac{\text{ft}^2}{144\text{ in.}^2} \times \frac{\text{lbf}\cdot\text{s}^2}{\text{slug}\cdot\text{ft}}$$

$$p_A - p_B = 3.73\text{ lbf/in.}^2 \longleftarrow p_A - p_B$$

This Example shows use of both Eq. 3.7 and Eq. 3.8. Use of either equation is a matter of personal preference.

Atmospheric pressure may be obtained from a *barometer*, in which the height of a mercury column is measured. The measured height may be converted to pressure using Eq. 3.7 and the data for specific gravity of mercury given in Appendix A, as discussed in the Notes of Example 3.1. Although the vapor pressure of mercury may be neglected, for precise work, temperature and altitude corrections must be applied to the measured level and the effects of surface tension must be considered. The capillary effect in a tube caused by surface tension was illustrated in Example 2.3.

Gases

In many practical engineering problems density will vary appreciably with altitude, and accurate results will require that this variation be accounted for. Pressure variation in a compressible fluid can be evaluated by integrating Eq. 3.6 if the density can be expressed as a function of p or z. Property information or an equation of state may be used to obtain the required relation for density. Several types of property variation may be analyzed. (See Example 3.4.)

The density of gases generally depends on pressure and temperature. The ideal gas equation of state,

$$p = \rho RT \tag{1.1}$$

where R is the gas constant (see Appendix A) and T the absolute temperature, accurately models the behavior of most gases under engineering conditions. However, the use of Eq. 1.1 introduces the gas temperature as an additional variable. Therefore, an additional assumption must be made about temperature variation before Eq. 3.6 can be integrated.

In the U.S. Standard Atmosphere the temperature decreases linearly with altitude up to an elevation of 11.0 km. For a linear temperature variation with altitude given by $T = T_0 - mz$, we obtain, from Eq. 3.6,

$$dp = -\rho g\, dz = -\frac{pg}{RT}dz = -\frac{pg}{R(T_0 - mz)}dz$$

Separating variables and integrating from $z = 0$ where $p = p_0$ to elevation z where the pressure is p gives

$$\int_{p_0}^{p} \frac{dp}{p} = -\int_0^z \frac{g dz}{R(T_0 - mz)}$$

Then

$$\ln\frac{p}{p_0} = \frac{g}{mR}\ln\left(\frac{T_0 - mz}{T_0}\right) = \frac{g}{mR}\ln\left(1 - \frac{mz}{T_0}\right)$$

and the pressure variation, in a gas whose temperature varies linearly with elevation, is given by

$$p = p_0\left(1 - \frac{mz}{T_0}\right)^{g/mR} = p_0\left(\frac{T}{T_0}\right)^{g/mR} \tag{3.9}$$

Example 3.4 PRESSURE AND DENSITY VARIATION IN THE ATMOSPHERE

The maximum power output capability of a gasoline or diesel engine decreases with altitude because the air density and hence the mass flow rate of air decrease. A truck leaves Denver (elevation 5280 ft) on a day when the local temperature and barometric pressure are 80°F and 24.8 in. of mercury, respectively. It travels through Vail Pass (elevation 10,600 ft), where the temperature is 62°F. Determine the local barometric pressure at Vail Pass and the percent change in density.

Given: Truck travels from Denver to Vail Pass.

Denver: $z = 5280$ ft, $p = 24.8$ in. Hg, $T = 80°$F

Vail Pass: $z = 10{,}600$ ft, $T = 62°$F

Find: Atmospheric pressure at Vail Pass.
Percent change in air density between Denver and Vail.

Solution:

Governing equations: $\dfrac{dp}{dz} = -\rho g \qquad p = \rho RT$

Assumptions: (1) Static fluid.
(2) Air behaves as an ideal gas.

We shall consider four assumptions for property variations with altitude.

(a) If we assume temperature varies linearly with altitude, Eq. 3.9 gives

$$\frac{p}{p_0} = \left(\frac{T}{T_0}\right)^{g/mR}$$

Evaluating the constant m gives

$$m = \frac{T_0 - T}{z - z_0} = \frac{(80 - 62)^\circ\text{F}}{(10.6 - 5.28)10^3\ \text{ft}} = 3.38 \times 10^{-3}\ ^\circ\text{F/ft}$$

and

$$\frac{g}{mR} = 32.2\frac{\text{ft}}{\text{s}^2} \times \frac{\text{ft}}{3.38 \times 10^{-3}\,°\text{F}} \times \frac{\text{lbm}\cdot°\text{R}}{53.3\,\text{ft}\cdot\text{lbf}} \times \frac{\text{slug}}{32.2\,\text{lbm}} \times \frac{\text{lbf}\cdot\text{s}^2}{\text{slug}\cdot\text{ft}} = 5.55$$

Thus

$$\frac{p}{p_0} = \left(\frac{T}{T_0}\right)^{g/mR} = \left(\frac{460+62}{460+80}\right)^{5.55} = (0.967)^{5.55} = 0.830$$

and

$$p = 0.830\,p_0 = (0.830)24.8\,\text{in. Hg} = 20.6\,\text{in. Hg} \xleftarrow{\qquad\qquad p \qquad\qquad}$$

Note that temperature must be expressed as an absolute temperature in the ideal gas equation of state.

The percent change in density is given by

$$\frac{\rho-\rho_0}{\rho_0} = \frac{\rho}{\rho_0} - 1 = \frac{p}{p_0}\frac{T_0}{T} - 1 = \frac{0.830}{0.967} - 1 = -0.142 \quad \text{or} \quad -14.2\% \xleftarrow{\qquad\qquad \frac{\Delta\rho}{\rho_0} \qquad\qquad}$$

(b) For ρ assumed constant $(=\rho_0)$,

$$p = p_0 - \rho_0 g(z-z_0) = p_0 - \frac{p_0 g\,(z-z_0)}{RT_0} = p_0\left[1 - \frac{g(z-z_0)}{RT_0}\right]$$

$$p = 20.2\,\text{in. Hg} \quad \text{and} \quad \frac{\Delta\rho}{\rho_0} = 0 \xleftarrow{\qquad\qquad p,\,\frac{\Delta\rho}{\rho_0} \qquad\qquad}$$

(c) If we assume the temperature is constant, then

$$dp = -\rho g\,dz = -\frac{p}{RT}g\,dz$$

and

$$\int_{p_0}^{p}\frac{dp}{p} = -\int_{z_0}^{z}\frac{g}{RT}dz$$

$$p = p_0 \exp\left[\frac{-g(z-z_0)}{RT}\right]$$

For $T = \text{constant} = T_0$,

$$p = 20.6\,\text{in. Hg} \qquad \text{and} \qquad \frac{\Delta\rho}{\rho_0} = -16.9\% \xleftarrow{\qquad\qquad p,\,\frac{\Delta\rho}{\rho_0} \qquad\qquad}$$

(d) For an adiabatic atmosphere $p/\rho^k = \text{constant}$,

$$p = p_0\left(\frac{T}{T_0}\right)^{k/k-1} = 22.0\,\text{in. Hg} \quad \text{and} \quad \frac{\Delta\rho}{\rho_0} = -8.2\% \xleftarrow{\qquad p,\,\frac{\Delta\rho}{\rho_0} \qquad}$$

We note that over the modest change in elevation the predicted pressure is not strongly dependent on the assumed property variation; values calculated under four different assumptions vary by a maximum of approximately 9 percent. There is considerably greater variation in the predicted percent change in density. The assumption of a linear temperature variation with altitude is the most reasonable assumption.

This Example shows use of the ideal gas equation with the basic pressure-height relation to obtain the change in pressure with height in the atmosphere under various atmospheric assumptions.

Hydraulic Systems 3.4

Hydraulic systems are characterized by very high pressures, so by comparison hydrostatic pressure variations often may be neglected. Automobile hydraulic brakes develop pressures up to 10 MPa (1500 psi); aircraft and machinery hydraulic actuation systems frequently are designed for pressures up to 40 MPa (6000 psi), and jacks use pressures to 70 MPa (10,000 psi). Special-purpose laboratory test equipment is commercially available for use at pressures to 1000 MPa (150,000 psi)!

Although liquids are generally considered incompressible at ordinary pressures, density changes may be appreciable at high pressures. Bulk moduli of hydraulic fluids also may vary sharply at high pressures. In problems involving unsteady flow, both compressibility of the fluid and elasticity of the boundary structure (e.g., the pipe walls) must be considered. Analysis of problems such as water hammer noise and vibration in hydraulic systems, actuators, and shock absorbers quickly becomes complex and is beyond the scope of this book.

VIDEO

Hydraulic Force Amplification.

Hydrostatic Force on Submerged Surfaces 3.5

Now that we have determined how the pressure varies in a static fluid, we can examine the force on a surface submerged in a liquid.

In order to determine completely the resultant force acting on a submerged surface, we must specify:

1. The magnitude of the force.
2. The direction of the force.
3. The line of action of the force.

We shall consider both plane and curved submerged surfaces.

Hydrostatic Force on a Plane Submerged Surface

A plane submerged surface, on whose upper face we wish to determine the resultant hydrostatic force, is shown in Fig. 3.5. The coordinates are important: They have been chosen so that the surface lies in the xy plane, and the origin O is located at the intersection of the plane surface (or its extension) and the free surface. As well as

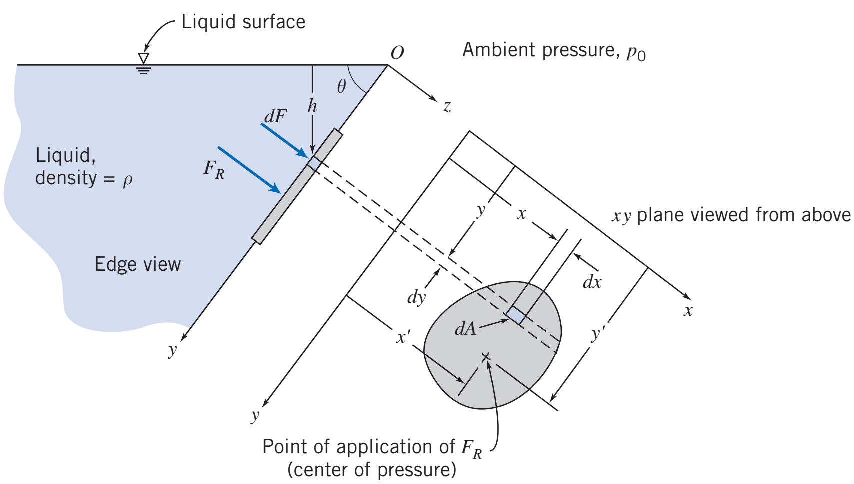


Fig. 3.5 Plane submerged surface.

the magnitude of the force F_R, we wish to locate the point (with coordinates x', y') through which it acts on the surface.

Since there are no shear stresses in a static fluid, the hydrostatic force on any element of the surface acts normal to the surface. The pressure force acting on an element $dA = dx\, dy$ of the upper surface is given by

$$dF = p\, dA$$

The *resultant* force acting on the surface is found by summing the contributions of the infinitesimal forces over the entire area.

Usually when we sum forces we must do so in a vectorial sense. However, in this case all of the infinitesimal forces are perpendicular to the plane, and hence so is the resultant force. Its magnitude is given by

$$F_R = \int_A p\, dA \tag{3.10a}$$

In order to evaluate the integral in Eq. 3.10a, both the pressure, p, and the element of area, dA, must be expressed in terms of the same variables.

We can use Eq. 3.7 to express the pressure p at depth h in the liquid as

$$p = p_0 + \rho g h$$

In this expression p_0 is the pressure at the free surface ($h = 0$).

In addition, we have, from the system geometry, $h = y \sin\theta$. Using this expression and the above expression for pressure in Eq. 3.10a,

$$F_R = \int_A p\, dA = \int_A (p_0 + \rho g h) dA = \int_A (p_0 + \rho g y \sin\theta) dA$$

$$F_R = p_0 \int_A dA + \rho g \sin\theta \int_A y\, dA = p_0 A + \rho g \sin\theta \int_A y\, dA$$

The integral is the first moment of the surface area about the x axis, which may be written

$$\int_A y\, dA = y_c A$$

where y_c is the y coordinate of the *centroid* of the area, A. Thus,

$$F_R = p_0 A + \rho g \sin\theta\, y_c A = (p_0 + \rho g h_c) A$$

or

$$F_R = p_c A \tag{3.10b}$$

where p_c is the absolute pressure in the liquid at the location of the centroid of area A. Equation 3.10b computes the resultant force due to the liquid—including the effect of the ambient pressure p_0—on one side of a submerged plane surface. It does not take into account whatever pressure or force distribution may be on the other side of the surface. However, if we have the *same* pressure, p_0, on this side as we do at the free surface of the liquid, as shown in Fig. 3.6, its effect on F_R cancels out, and if we wish to obtain the *net* force on the surface we can use Eq. 3.10b *with p_c expressed as a gage rather than absolute pressure*.

In computing F_R we can use either the integral of Eq. 3.10a or the resulting Eq. 3.10b. It is important to note that even though the force can be computed using the pressure at the center of the plate, this is *not* the point through which the force acts!

Our next task is to determine (x', y'), the location of the resultant force. Let's first obtain y' by recognizing that the moment of the resultant force about the x axis must

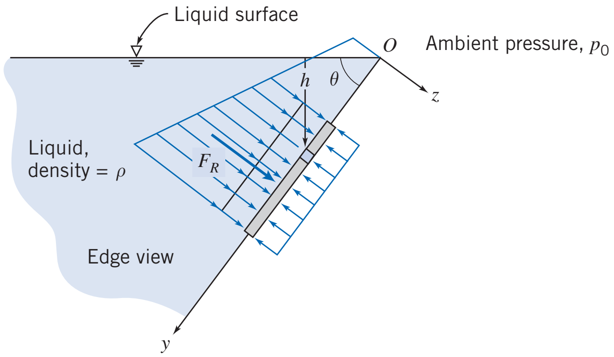


Fig. 3.6 Pressure distribution on plane submerged surface.

be equal to the moment due to the distributed pressure force. Taking the sum (i.e., integral) of the moments of the infinitesimal forces dF about the x axis we obtain

$$y' F_R = \int_A yp\, dA \tag{3.11a}$$

We can integrate by expressing p as a function of y as before:

$$\begin{aligned} y'F_R &= \int_A yp\, dA = \int_A y(p_0 + \rho g h)\, dA = \int_A (p_0 y + \rho g y^2 \sin\theta)\, dA \\ &= p_0 \int_A y\, dA + \rho g \sin\theta \int_A y^2 dA \end{aligned}$$

The first integral is our familiar $y_c A$. The second integral, $\int_A y^2\, dA$, is the second moment of area about the x axis, I_{xx}. We can use the parallel axis theorem, $I_{xx} = I_{\hat{x}\hat{x}} + Ay_c^2$, to replace I_{xx} with the standard second moment of area, about the centroidal $\hat{x}$ axis. Using all of these, we find

$$\begin{aligned} y'F_R &= p_0 y_c A + \rho g \sin\theta (I_{\hat{x}\hat{x}} + Ay_c^2) = y_c(p_0 + \rho g y_c \sin\theta)A + \rho g \sin\theta\, I_{\hat{x}\hat{x}} \\ &= y_c(p_0 + \rho g h_c)A + \rho g \sin\theta I_{\hat{x}\hat{x}} = y_c F_R + \rho g \sin\theta I_{\hat{x}\hat{x}} \end{aligned}$$

Finally, we obtain for y':

$$y' = y_c + \frac{\rho g \sin\theta\, I_{\hat{x}\hat{x}}}{F_R} \tag{3.11b}$$

Equation 3.11b is convenient for computing the location y' of the force on the submerged side of the surface when we include the ambient pressure p_0. If we have the same ambient pressure acting on the other side of the surface we can use Eq. 3.10b with p_0 neglected to compute the net force,

$$F_R = p_{c_{\text{gage}}} A = \rho g h_c A = \rho g y_c \sin\theta\, A$$

and Eq. 3.11b becomes for this case

$$y' = y_c + \frac{I_{\hat{x}\hat{x}}}{Ay_c} \tag{3.11c}$$

Equation 3.11a is the integral equation for computing the location y' of the resultant force; Eq. 3.11b is a useful algebraic form for computing y' when we are interested in the resultant force on the submerged side of the surface; Eq. 3.11c is for computing y' when we are interested in the net force for the case when the same p_0 acts at the free surface and on the other side of the submerged surface. For problems that have a pressure on the other side that is *not* p_0, we can either analyze each side of the surface separately or reduce the two pressure distributions to one net pressure distribution, in effect creating a system to be solved using Eq. 3.10b with p_c expressed as a gage pressure.

Note that in any event, $y' > y_c$—the location of the force is always below the level of the plate centroid. This makes sense—as Fig. 3.6 shows, the pressures will always be larger on the lower regions, moving the resultant force down the plate.

A similar analysis can be done to compute x', the x location of the force on the plate. Taking the sum of the moments of the infinitesimal forces dF about the y axis we obtain

$$x' F_R = \int_A x\,p\,dA \tag{3.12a}$$

We can express p as a function of y as before:

$$\begin{aligned} x' F_R &= \int_A xp\,dA = \int_A x(p_0 + \rho g h)\,dA = \int_A (p_0 x + \rho g x y \sin\theta)\,dA \\ &= p_0 \int_A x\,dA + \rho g \sin\theta \int_A xy\,dA \end{aligned}$$

The first integral is $x_c A$ (where x_c is the distance of the centroid from y axis). The second integral is $\int_A xy\,dA = I_{xy}$. Using the parallel axis theorem, $I_{xy} = I_{\hat{x}\hat{y}} + A x_c y_c$, we find

$$\begin{aligned} x' F_R &= p_0 x_c A + \rho g \sin\theta (I_{\hat{x}\hat{y}} + A x_c y_c) = x_c(p_0 + \rho g y_c \sin\theta) A + \rho g \sin\theta\, I_{\hat{x}\hat{y}} \\ &= x_c(p_0 + \rho g h_c) A + \rho g \sin\theta\, I_{\hat{x}\hat{y}} = x_c F_R + \rho g \sin\theta\, I_{\hat{x}\hat{y}} \end{aligned}$$

Finally, we obtain for x':

$$x' = x_c + \frac{\rho g \sin\theta\, I_{\hat{x}\hat{y}}}{F_R} \tag{3.12b}$$

Equation 3.12b is convenient for computing x' when we include the ambient pressure p_0. If we have ambient pressure also acting on the other side of the surface we can again use Eq. 3.10b with p_0 neglected to compute the net force and Eq. 3.12b becomes for this case

$$x' = x_c + \frac{I_{\hat{x}\hat{y}}}{A y_c} \tag{3.12c}$$

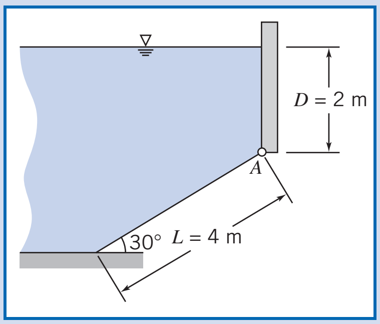
Equation 3.12a is the integral equation for computing the location x' of the resultant force; Eq. 3.12b can be used for computations when we are interested in the force on the submerged side only; Eq. 3.12c is useful when we have p_0 on the other side of the surface and we are interested in the net force.

In summary, Eqs. 3.10 through 3.12 constitute a complete set of equations for computing the magnitude and location of the force due to hydrostatic pressure on any submerged plane surface. The direction of the force will always be perpendicular to the plane.

We can now consider several examples using these equations. In Example 3.5 we use both the integral and algebraic sets of equations.

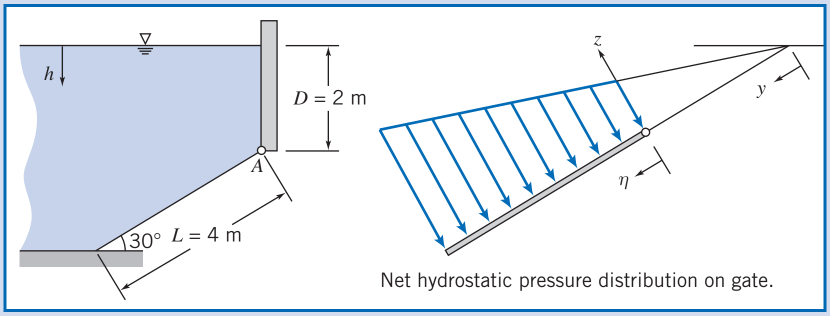
Example 3.5 RESULTANT FORCE ON INCLINED PLANE SUBMERGED SURFACE

The inclined surface shown, hinged along edge A, is 5 m wide. Determine the resultant force, F_R, of the water and the air on the inclined surface.



Given: Rectangular gate, hinged along A, $w = 5$ m.

Find: Resultant force, F_R, of the water and the air on the gate.



Net hydrostatic pressure distribution on gate.

Solution:

In order to completely determine F_R, we need to find (a) the magnitude and (b) the line of action of the force (the direction of the force is perpendicular to the surface). We will solve this problem by using (i) direct integration and (ii) the algebraic equations.

Direct Integration

Governing equations: $p = p_0 + \rho g h \quad F_R = \int_A p\, dA \quad \eta' F_R = \int_A \eta p\, dA \quad x' F_R = \int_A x p\, dA$

Because atmospheric pressure p_0 acts on both sides of the plate its effect cancels, and we can work in gage pressures ($p = \rho g h$). In addition, while we *could* integrate using the y variable, it will be more convenient here to define a variable η, as shown in the figure.

Using η to obtain expressions for h and dA, then

$$h = D + \eta \sin 30^\circ \qquad \text{and} \qquad dA = w\, d\eta$$

Applying these to the governing equation for the resultant force,

$$F_R = \int_A p dA = \int_0^L \rho g (D + \eta \sin 30^\circ) w\, d\eta$$

$$= \rho g w \left[D\eta + \frac{\eta^2}{2} \sin 30^\circ \right]_0^L = \rho g w \left[DL + \frac{L^2}{2} \sin 30^\circ \right]$$

$$= 999 \frac{\text{kg}}{\text{m}^3} \times 9.81 \frac{\text{m}}{\text{s}^2} \times 5\text{m} \left[2\,\text{m} \times 4\,\text{m} + \frac{16\,\text{m}^2}{2} \times \frac{1}{2} \right] \frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}}$$

$$F_R = 588\,\text{kN} \longleftarrow F_R$$

For the location of the force we compute η' (the distance from the top edge of the plate),

$$\eta' F_R = \int_A \eta p\, dA$$

Then

$$\eta' = \frac{1}{F_R} \int_A \eta p dA = \frac{1}{F_R} \int_0^L \eta p w\, d\eta = \frac{\rho g w}{F_R} \int_0^L \eta (D + \eta \sin 30^\circ)\, d\eta$$

$$= \frac{\rho g w}{F_R} \left[\frac{D\eta^2}{2} + \frac{\eta^3}{3} \sin 30^\circ \right]_0^L = \frac{\rho g w}{F_R} \left[\frac{DL^2}{2} + \frac{L^3}{3} \sin 30^\circ \right]$$

$$= 999 \frac{\text{kg}}{\text{m}^3} \times 9.8 \frac{\text{m}}{\text{s}^2} \times \frac{5\,\text{m}}{5.88 \times 10^5\,\text{N}} \left[\frac{2\,\text{m} \times 16\,\text{m}^2}{2} + \frac{64\,\text{m}^3}{3} \times \frac{1}{2} \right] \frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}}$$

$$\eta' = 2.22\,\text{m} \quad \text{and} \quad y' = \frac{D}{\sin 30^\circ} + \eta' = \frac{2\,\text{m}}{\sin 30^\circ} + 2.22\,\text{m} = 6.22\,\text{m} \longleftarrow y'$$

Also, from consideration of moments about the y axis through edge A,

$$x' = \frac{1}{F_R}\int_A xp\,dA$$

In calculating the moment of the distributed force (right side), recall, from your earlier courses in statics, that the centroid of the area element must be used for x. Since the area element is of constant width, then $x = w/2$, and

$$x' = \frac{1}{F_R}\int_A \frac{w}{2}p\,dA = \frac{w}{2F_R}\int_A p\,dA = \frac{w}{2} = 2.5\text{ m} \xleftarrow{\qquad\qquad x' \qquad\qquad}$$

Algebraic Equations

In using the algebraic equations we need to take care in selecting the appropriate set. In this problem we have $p_0 = p_{\text{atm}}$ on both sides of the plate, so Eq. 3.10b with p_c as a gage pressure is used for the net force:

$$F_R = p_c A = \rho g h_c A = \rho g\left(D + \frac{L}{2}\sin 30^\circ\right)Lw$$

$$F_R = \rho g w\left[DL + \frac{L^2}{2}\sin 30^\circ\right]$$

This is the same expression as was obtained by direct integration.

The y coordinate of the center of pressure is given by Eq. 3.11c:

$$y' = y_c + \frac{I_{\hat{x}\hat{x}}}{Ay_c} \tag{3.11c}$$

For the inclined rectangular gate

$$y_c = \frac{D}{\sin 30^\circ} + \frac{L}{2} = \frac{2\text{ m}}{\sin 30^\circ} + \frac{4\text{ m}}{2} = 6\text{ m}$$

$$A = Lw = 4\text{ m} \times 5\text{ m} = 20\text{ m}^2$$

$$I_{\hat{x}\hat{x}} = \frac{1}{12}wL^3 = \frac{1}{12} \times 5\text{ m} \times (4\text{ m})^3 = 26.7\text{ m}^2$$

$$y' = y_c + \frac{I_{\hat{x}\hat{x}}}{Ay_c} = 6\text{ m} + 26.7\text{ m}^4 \times \frac{1}{20\text{ m}^2} \times \frac{1}{6\text{ m}} = 6.22\text{ m} \xleftarrow{\qquad\qquad y' \qquad\qquad}$$

The x coordinate of the center of pressure is given by Eq. 3.12c:

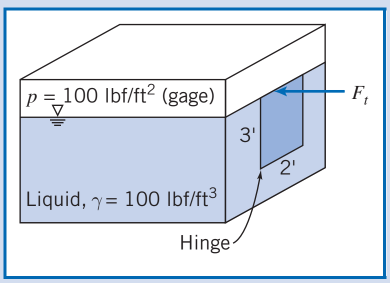
$$x' = x_c + \frac{I_{\hat{x}\hat{y}}}{Ay_c} \tag{3.12c}$$

For the rectangular gate $I_{\hat{x}\hat{y}} = 0$ and $x' = x_c = 2.5$ m. $\xleftarrow{\qquad\qquad x' \qquad\qquad}$

This Example shows
- ✓ Use of integral and algebraic equations.
- ✓ Use of the algebraic equations for computing the *net* force.

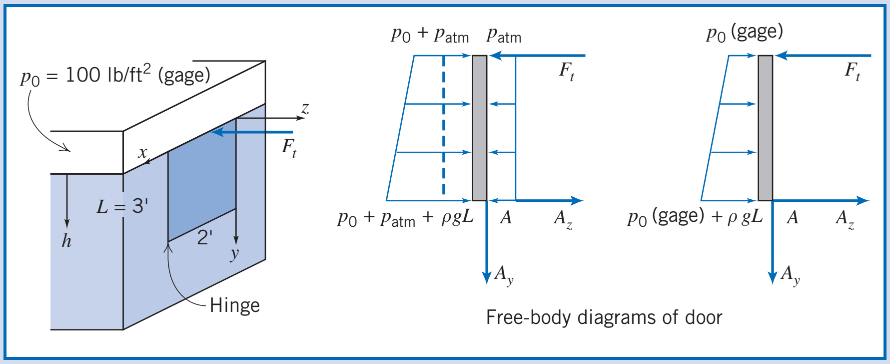
Example 3.6 FORCE ON VERTICAL PLANE SUBMERGED SURFACE WITH NONZERO GAGE PRESSURE AT FREE SURFACE

The door shown in the side of the tank is hinged along its bottom edge. A pressure of 100 psfg is applied to the liquid free surface. Find the force, F_t, required to keep the door closed.

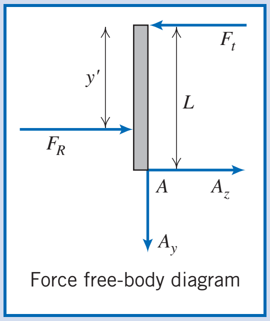


Given: Door as shown in the figure.

Find: Force required to keep door shut.



Free-body diagrams of door



Force free-body diagram

Solution:

This problem requires a free-body diagram (FBD) of the door. The pressure distributions on the inside and outside of the door will lead to a net force (and its location) that will be included in the FBD. We need to be careful in choosing the equations for computing the resultant force and its location. We can either use absolute pressures (as on the left FBD) and compute two forces (one on each side) or gage pressures and compute one force (as on the right FBD). For simplicity we will use gage pressures. The right-hand FBD makes clear we should use Eqs. 3.10b and 3.11b, which were derived for problems in which we wish to include thc effects of an ambient pressure (p_0), or in other words, for problems when we have a nonzero gage pressure at the free surface. The components of force due to the hinge are A_y and A_z. The force F_t can be found by taking moments about A (the hinge).

Governing equations:

$$F_R = p_c A \qquad y' = y_c + \frac{\rho g \sin\theta \, I_{\hat{x}\hat{x}}}{F_R} \qquad \sum M_A = 0$$

The resultant force and its location are

$$F_R = (p_0 + \rho g h_c)A = \left(p_0 + \gamma \frac{L}{2}\right) bL \tag{1}$$

and

$$y' = y_c + \frac{\rho g \sin 90^\circ \, I_{\hat{x}\hat{x}}}{F_R} = \frac{L}{2} + \frac{\gamma b L^3/12}{\left(p_0 + \gamma \frac{L}{2}\right) bL} = \frac{L}{2} + \frac{\gamma L^2/12}{\left(p_0 + \gamma \frac{L}{2}\right)} \tag{2}$$

Taking moments about point A

$$\sum M_A = F_t L - F_R(L - y') = 0 \qquad \text{or} \qquad F_t = F_R\left(1 - \frac{y'}{L}\right)$$

Using Eqs. 1 and 2 in this equation we find

$$F_t = \left(p_0 + \gamma \frac{L}{2}\right) bL \left[1 - \frac{1}{2} - \frac{\gamma L^2/12}{\left(p_0 + \gamma \frac{L}{2}\right)}\right]$$

$$\begin{aligned} F_t &= \left(p_0 + \gamma \frac{L}{2}\right) \frac{bL}{2} + \gamma \frac{bL^2}{12} = \frac{p_0 bL}{2} + \frac{\gamma b L^2}{6} \\ &= 100 \,\frac{\text{lbf}}{\text{ft}^2} \times 2\,\text{ft} \times 3\,\text{ft} \times \frac{1}{2} + 100\,\frac{\text{lbf}}{\text{ft}^3} \times 2\,\text{ft} \times 9\,\text{ft}^2 \times \frac{1}{6} \end{aligned} \tag{3}$$

$$F_t = 600\,\text{lbf} \xleftarrow{\hspace{6em} F_t}$$

We could have solved this problem by considering the two separate pressure distributions on each side of the door, leading to two resultant forces and their locations. Summing moments about point A with these forces would also have yielded the same value for F_t. (See Problem 3.59.) Note also that Eq. 3 could have been obtained directly (without separately finding F_R and y') by using a direct integration approach:

$$\sum M_A = F_t L - \int_A y\, p\, dA = 0$$

This Example shows:

✓ Use of algebraic equations for nonzero gage pressure at the liquid free surface.

✓ Use of the moment equation from statics for computing the required applied force.

Hydrostatic Force on a Curved Submerged Surface

For curved surfaces, we will once again derive expressions for the resultant force by integrating the pressure distribution over the surface. However, unlike for the plane surface, we have a more complicated problem—the pressure force is normal to the surface at each point, but now the infinitesimal area elements point in varying directions because of the surface curvature. This means that instead of integrating over an element dA we need to integrate over vector element $d\vec{A}$. This will initially lead to a more complicated analysis, but we will see that a simple solution technique will be developed.

Consider the curved surface shown in Fig. 3.7. The pressure force acting on the element of area, $d\vec{A}$, is given by

$$d\vec{F} = -p\, d\vec{A}$$

where the minus sign indicates that the force acts on the area, in the direction opposite to the area normal. The resultant force is given by

$$\vec{F}_R = -\int_A p\, d\vec{A} \tag{3.13}$$

We can write

$$\vec{F}_R = \hat{i}F_{R_x} + \hat{j}F_{R_y} + \hat{k}F_{R_z}$$

where F_{Rx}, F_{Ry}, and F_{Rz} are the components of $\vec{F}_R$ in the positive x, y, and z directions, respectively.

To evaluate the component of the force in a given direction, we take the dot product of the force with the unit vector in the given direction. For example, taking the dot product of each side of Eq. 3.13 with unit vector $\hat{i}$ gives

$$F_{R_x} = \vec{F}_R \cdot \hat{i} = \int d\vec{F}\cdot\hat{i} = -\int_A p\, d\vec{A}\cdot\hat{i} = -\int_{A_x} p\, dA_x$$

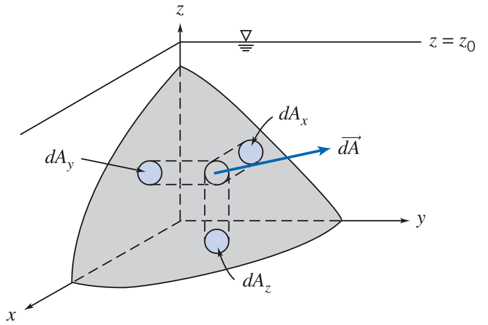


Fig. 3.7 Curved submerged surface.

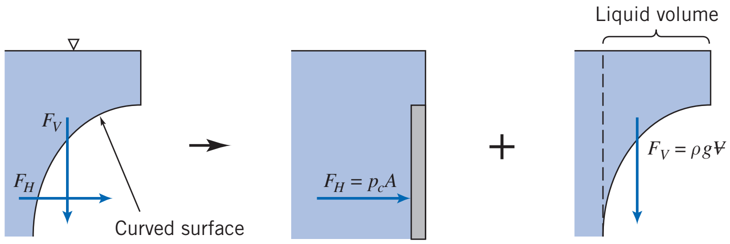


Fig. 3.8 Forces on curved submerged surface.

where dA_x is the projection of $d\vec{A}$ on a plane perpendicular to the x axis (see Fig. 3.7), and the minus sign indicates that the x component of the resultant force is in the negative x direction.

Since, in any problem, the direction of the force component can be determined by inspection, the use of vectors is not necessary. In general, the magnitude of the component of the resultant force in the l direction is given by

$$F_{R_l} = \int_{A_l} p\, dA_l \tag{3.14}$$

where dA_l is the projection of the area element dA on a plane perpendicular to the l direction. The line of action of each component of the resultant force is found by recognizing that the moment of the resultant force component about a given axis must be equal to the moment of the corresponding distributed force component about the same axis.

Equation 3.14 can be used for the horizontal forces F_{R_x} and F_{R_y}. We have the interesting result that *the horizontal force and its location are the same as for an imaginary vertical plane surface of the same projected area*. This is illustrated in Fig. 3.8, where we have called the horizontal force F_H.

Figure 3.8 also illustrates how we can compute the vertical component of force: With atmospheric pressure at the free surface and on the other side of the curved surface *the net vertical force will be equal to the weight of fluid directly above the surface*. This can be seen by applying Eq. 3.14 to determine the magnitude of the vertical component of the resultant force, obtaining

$$F_{R_z} = F_V = \int p\, dA_z$$

Since $p = \rho g h$,

$$F_V = \int \rho g h\, dA_z = \int \rho g\, dV\!\!\!\!-$$

where $\rho g h\, dA_z = \rho g\, dV\!\!\!\!-$ is the weight of a differential cylinder of liquid above the element of surface area, dA_z, extending a distance h from the curved surface to the free surface. The vertical component of the resultant force is obtained by integrating over the entire submerged surface. Thus

$$F_V = \int_{A_z} \rho g h\, dA_z = \int_{V\!\!\!\!-} \rho g\, dV\!\!\!\!- = \rho g\, V\!\!\!\!-$$

In summary, for a curved surface we can use two simple formulas for computing the horizontal and vertical force components due to the fluid only (no ambient pressure),

$$F_H = p_c A \quad \text{and} \quad F_V = \rho g V\!\!\!\!- \tag{3.15}$$

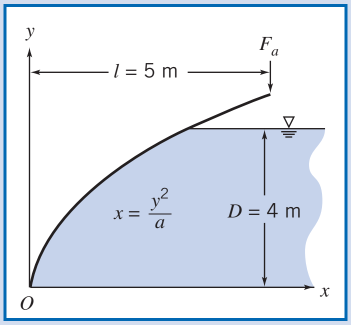
where p_c and A are the pressure at the center and the area, respectively, of a vertical plane surface of the same projected area, and $\forall$ is the volume of fluid above the curved surface.

It can be shown that the line of action of the vertical force component passes through the center of gravity of the volume of liquid directly above the curved surface (see Example 3.7).

We have shown that the resultant hydrostatic force on a curved submerged surface is specified in terms of its components. We recall from our study of statics that the resultant of any force system can be represented by a force-couple system, i.e., the resultant force applied at a point and a couple about that point. If the force and the couple vectors are orthogonal (as is the case for a two-dimensional curved surface), the resultant can be represented as a pure force with a unique line of action. Otherwise the resultant may be represented as a "wrench," also having a unique line of action.

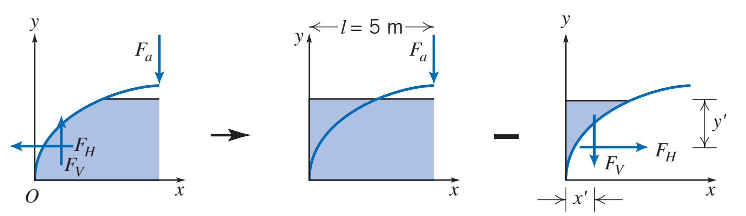
Example 3.7 FORCE COMPONENTS ON A CURVED SUBMERGED SURFACE

The gate shown is hinged at O and has constant width, $w = 5$ m. The equation of the surface is $x = y^2/a$, where $a = 4$ m. The depth of water to the right of the gate is $D = 4$ m. Find the magnitude of the force, F_a, applied as shown, required to maintain the gate in equilibrium if the weight of the gate is neglected.



Given: Gate of constant width, $w = 5$ m.
Equation of surface in xy plane is $x = y^2/a$, where $a = 4$ m.
Water stands at depth $D = 4$ m to the right of the gate.
Force F_a is applied as shown, and weight of gate is to be neglected. (Note that for simplicity we do not show the reactions at O.)

Find: Force F_a required to maintain the gate in equilibrium.



(*a*) System FBD (*b*) Null fluid forces (*c*) Fluid forces

Solution:

We will take moments about point O after finding the magnitudes and locations of the horizontal and vertical forces due to the water. The free body diagram (FBD) of the system is shown above in part (a). Before proceeding we need to think about how we compute F_V, the vertical component of the fluid force—we have stated that it is equal (in magnitude and location) to the weight of fluid directly above the curved surface. However, we have no fluid directly above the gate, even though it is clear that the fluid does exert a vertical force! We need to do a "thought experiment" in which we imagine having a system with water on both sides of the gate (with null effect), minus a system with water directly above the gate (which generates fluid forces). This logic is demonstrated above: the system FBD(a) = the null FBD(b) − the fluid forces FBD(c). Thus the vertical and horizontal fluid forces on the system, FBD(a), are equal and opposite to those on FBD(c). In summary, the magnitude and location of the vertical fluid force F_V are given by the weight and location of the centroid of the fluid "above" the gate; the magnitude and location of the horizontal fluid force F_H are given by the magnitude and location of the force on an equivalent vertical flat plate.

Governing equations: $F_H = p_c A \qquad y' = y_c + \frac{I_{\hat{x}\hat{x}}}{Ay_c} \qquad F_V = \rho g V\!\!\!/ \qquad x' =$ water center of gravity

For F_H, the centroid, area, and second moment of the equivalent vertical flat plate are, respectively, $y_c = h_c = D/2$, $A = Dw$, and $I_{\hat{x}\hat{x}} = wD^3/12$.

$$\begin{aligned} F_H &= p_c A = \rho g h_c A \\ &= \rho g \frac{D}{2} Dw = \rho g \frac{D^2}{2} w = 999 \frac{\text{kg}}{\text{m}^3} \times 9.81 \frac{\text{m}}{\text{s}^2} \times \frac{(4\,\text{m}^2)}{2} \times 5\,\text{m} \times \frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}} \qquad (1) \\ F_H &= 392\,\text{kN} \end{aligned}$$

and

$$\begin{aligned} y' &= y_c + \frac{I_{\hat{x}\hat{x}}}{Ay_c} \\ &= \frac{D}{2} + \frac{wD^3/12}{wDD/2} = \frac{D}{2} + \frac{D}{6} \\ y' &= \frac{2}{3}D = \frac{2}{3} \times 4\,\text{m} = 2.67\,\text{m} \qquad (2) \end{aligned}$$

For F_V, we need to compute the weight of water "above" the gate. To do this we define a differential column of volume $(D - y)\ w\ dx$ and integrate

$$\begin{aligned} F_V &= \rho g V\!\!\!/ = \rho g \int_0^{D^{2/a}} (D-y) w\, dx = \rho g w \int_0^{D^{2/a}} (D - \sqrt{a} x^{1/2}) dx \\ &= \rho g w \left[Dx - \frac{2}{3}\sqrt{a} x^{3/2} \right]_0^{D^{3/a}} = \rho g w \left[\frac{D^3}{a} - \frac{2}{3}\sqrt{a}\frac{D^3}{a^{3/2}} \right] = \frac{\rho g w D^3}{3a} \\ F_V &= 999 \frac{\text{kg}}{\text{m}^3} \times 9.81 \frac{\text{m}}{\text{s}^2} \times 5\,\text{m} \times \frac{(4)^3 \text{m}^3}{3} \times \frac{1}{4\,\text{m}} \times \frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}} = 261\,\text{kN} \qquad (3) \end{aligned}$$

The location x' of this force is given by the location of the center of gravity of the water "above" the gate. We recall from statics that this can be obtained by using the notion that the moment of F_V and the moment of the sum of the differential weights about the y axis must be equal, so

$$\begin{aligned} x'F_V &= \rho g \int_0^{D^{2/a}} x(D-y) w\, dx = \rho g w \int_0^{D^{2/a}} (D - \sqrt{a} x^{3/2}) dx \\ x'F_V &= \rho g w \left[\frac{D}{2} x^2 - \frac{2}{5}\sqrt{a} x^{5/2} \right]_0^{D^{2/a}} = \rho g w \left[\frac{D^5}{2a^2} - \frac{2}{5}\sqrt{a}\frac{D^5}{a^{5/2}} \right] = \frac{\rho g w D^5}{10a^2} \\ x' &= \frac{\rho g w D^5}{10a^2 F_V} = \frac{3D^2}{10a} = \frac{3}{10} \times \frac{(4)^2\,\text{m}^2}{4\,\text{m}} = 1.2\,\text{m} \qquad (4) \end{aligned}$$

Now that we have determined the fluid forces, we can finally take moments about O (taking care to use the appropriate signs), using the results of Eqs. 1 through 4

$$\begin{aligned} \sum M_O &= -lF_a + x'F_V + (D - y')F_H = 0 \\ F_a &= \frac{1}{l}[x'F_V + (D - y')F_H] \\ &= \frac{1}{5\,\text{m}}[1.2\,\text{m} \times 261\,\text{kN} + (4 - 2.67)\,\text{m} \times 392\,\text{kN}] \\ F_a &= 167\ \text{kN} \qquad \xleftarrow{\qquad F_a \qquad} \end{aligned}$$

This Example shows:

- ✓ Use of vertical flat plate equations for the horizontal force, and fluid weight equations for the vertical force, on a curved surface.
- ✓ The use of "thought experiments" to convert a problem with fluid below a curved surface into an equivalent problem with fluid above.

*3.6 Buoyancy and Stability

If an object is immersed in a liquid, or floating on its surface, the net vertical force acting on it due to liquid pressure is termed *buoyancy*. Consider an object totally immersed in static liquid, as shown in Fig. 3.9.

The vertical force on the body due to hydrostatic pressure may be found most easily by considering cylindrical volume elements similar to the one shown in Fig. 3.9.

We recall that we can use Eq. 3.7 for computing the pressure p at depth h in a liquid,

$$p = p_0 + \rho g h$$

The net vertical pressure force on the element is then

$$dF_z = (p_0 + \rho g h_2)\, dA - (p_0 + \rho g h_1)\, dA = \rho g (h_2 - h_1)\, dA$$

But $(h_2 - h_1)dA = d\forall$, the volume of the element. Thus

$$F_z = \int dF_z = \int_{\forall} \rho g d\forall = \rho g \forall$$

where $\forall$ is the volume of the object. Hence we conclude that for a submerged body *the buoyancy force of the fluid is equal to the weight of displaced fluid*,

$$F_{\text{buoyancy}} = \rho g \forall \tag{3.16}$$

This relation reportedly was used by Archimedes in 220 B.C. to determine the gold content in the crown of King Hiero II. Consequently, it is often called "Archimedes' Principle." In more current technical applications, Eq. 3.16 is used to design displacement vessels, flotation gear, and submersibles [1].

The submerged object need not be solid. Hydrogen bubbles, used to visualize streaklines and timelines in water (see Section 2.2), are positively buoyant; they rise slowly as they are swept along by the flow. Conversely, water droplets in oil are negatively buoyant and tend to sink.

Airships and balloons are termed "lighter-than-air" craft. The density of an ideal gas is proportional to molecular weight, so hydrogen and helium are less dense than air at the same temperature and pressure. Hydrogen ($M_m = 2$) is less dense than helium ($M_m = 4$), but extremely flammable, whereas helium is inert. Hydrogen has not been used commercially since the disastrous explosion of the German passenger airship *Hindenburg* in 1937. The use of buoyancy force to generate lift is illustrated in Example 3.8.

Equation 3.16 predicts the net vertical pressure force on a body that is totally submerged in a single liquid. In cases of partial immersion, a floating body displaces its own weight of the liquid in which it floats.

The line of action of the buoyancy force, which may be found using the methods of Section 3.5, acts through the centroid of the displaced volume. Since floating bodies

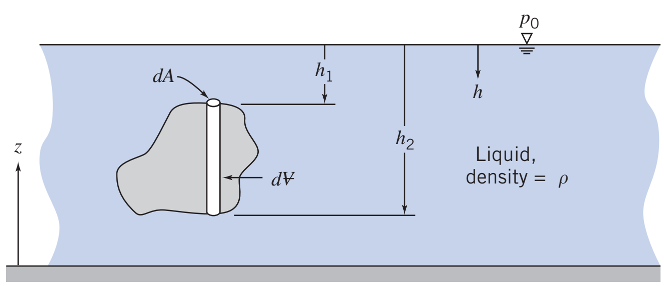
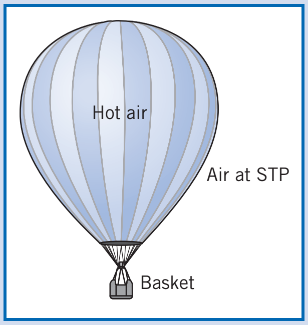


Fig. 3.9 Immersed body in static liquid.

[1]This section may be omitted without loss of continuity in the text material.

Example 3.8 BUOYANCY FORCE IN A HOT AIR BALLOON



A hot air balloon (approximated as a sphere of diameter 50 ft) is to lift a basket load of 600 lbf. To what temperature must the air be heated in order to achieve liftoff?

Given: Atmosphere at STP, diameter of balloon $d = 50$ ft, and load $W_{\text{load}} = 600$ lbf.

Find: The hot air temperature to attain liftoff.

Solution:
Apply the buoyancy equation to determine the lift generated by atmosphere, and apply the vertical force equilibrium equation to obtain the hot air density. Then use the ideal gas equation to obtain the hot air temperature.

Governing equations:

$$F_{\text{buoyancy}} = \rho g \forall \qquad \sum F_y = 0 \qquad p = \rho R T$$

Assumptions: (1) Ideal gas.
(2) Atmospheric pressure throughout.

Summing vertical forces

$$\sum F_y = F_{\text{buoyancy}} - W_{\text{hot air}} - W_{\text{load}} = \rho_{\text{atm}} g \forall - \rho_{\text{hot air}} g \forall - W_{\text{load}} = 0$$

Rearranging and solving for $\rho_{\text{hot air}}$ (using data from Appendix A),

$$\rho_{\text{hot air}} = \rho_{\text{atm}} - \frac{W_{\text{load}}}{g\forall} = \rho_{\text{atm}} - \frac{6W_{\text{load}}}{\pi d^3 g}$$

$$= 0.00238 \frac{\text{slug}}{\text{ft}^3} - 6 \times \frac{600\,\text{lbf}}{\pi(50)^3\,\text{ft}^3} \times \frac{\text{s}^2}{32.2\,\text{ft}} \times \frac{\text{slug}\cdot\text{ft}}{\text{s}^2\cdot\text{lbf}}$$

$$\rho_{\text{hot air}} = (0.00238 - 0.000285)\frac{\text{slug}}{\text{ft}^3} = 0.00209\frac{\text{slug}}{\text{ft}^3}$$

Finally, to obtain the temperature of this hot air, we can use the ideal gas equation in the following form

$$\frac{p_{\text{hot air}}}{\rho_{\text{hot air}} R T_{\text{hot air}}} = \frac{p_{\text{atm}}}{\rho_{\text{atm}} R T_{\text{atm}}}$$

and with $p_{\text{hot air}} = p_{\text{atm}}$

$$T_{\text{hot air}} = T_{\text{atm}} \frac{\rho_{\text{atm}}}{\rho_{\text{hot air}}} = (460 + 59)^\circ\text{R} \times \frac{0.00238}{0.00209} = 591^\circ\text{R}$$

$$T_{\text{hot air}} = 131^\circ\text{F} \longleftarrow T_{\text{hot air}}$$

Notes:
- ✓ Absolute pressures and temperatures are always used in the ideal gas equation.
- ✓ This problem demonstrates that for lighter-than-air vehicles the buoyancy force exceeds the vehicle weight—that is, the weight of fluid (air) displaced exceeds the vehicle weight.

are in equilibrium under body and buoyancy forces, the location of the line of action of the buoyancy force determines stability, as shown in Fig. 3.10.

The weight of an object acts through its center of gravity, CG. In Fig. 3.10*a*, the lines of action of the buoyancy and the weight are offset in such a way as to produce a couple that tends to right the craft. In Fig. 3.10*b*, the couple tends to capsize the craft.

Ballast may be needed to achieve roll stability. Wooden warships carried stone ballast low in the hull to offset the weight of the heavy cannon on upper gun decks.

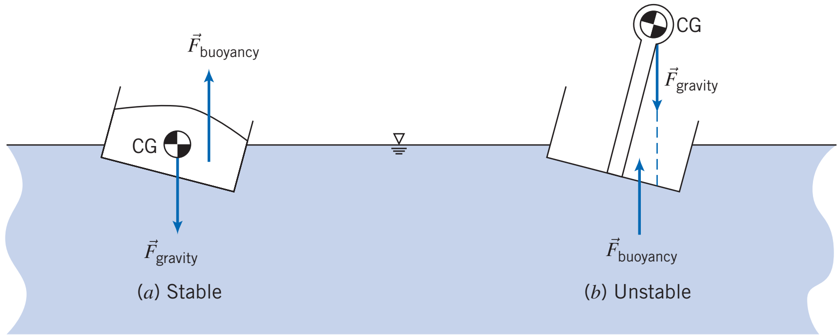


Fig. 3.10 Stability of floating bodies.

Modern ships can have stability problems as well: overloaded ferry boats have capsized when passengers all gathered on one side of the upper deck, shifting the CG laterally. In stacking containers high on the deck of a container ship, care is needed to avoid raising the center of gravity to a level that may result in the unstable condition depicted in Fig. 3.10*b*.

For a vessel with a relatively flat bottom, as shown in Fig. 3.10*a*, the restoring moment increases as roll angle becomes larger. At some angle, typically that at which the edge of the deck goes below water level, the restoring moment peaks and starts to decrease. The moment may become zero at some large roll angle, known as the angle of vanishing stability. The vessel may capsize if the roll exceeds this angle; then, if still intact, the vessel may find a new equilibrium state upside down.

The actual shape of the restoring moment curve depends on hull shape. A broad beam gives a large lateral shift in the line of action of the buoyancy force and thus a high restoring moment. High freeboard above the water line increases the angle at which the moment curve peaks, but may make the moment drop rapidly above this angle.

Sailing vessels are subjected to large lateral forces as wind engages the sails (a boat under sail in a brisk wind typically operates at a considerable roll angle). The lateral wind force must be counteracted by a heavily weighted keel extended below the hull bottom. In small sailboats, crew members may lean far over the side to add additional restoring moment to prevent capsizing [2].

Within broad limits, the buoyancy of a surface vessel is adjusted automatically as the vessel rides higher or lower in the water. However, craft that operate fully submerged must actively adjust buoyancy and gravity forces to remain neutrally buoyant. For submarines this is accomplished using tanks which are flooded to reduce excess buoyancy or blown out with compressed air to increase buoyancy [1]. Airships may vent gas to descend or drop ballast to rise. Buoyancy of a hot-air balloon is controlled by varying the air temperature within the balloon envelope.

For deep ocean dives use of compressed air becomes impractical because of the high pressures (the Pacific Ocean is over 10 km deep; seawater pressure at this depth is greater than 1000 atmospheres!). A liquid such as gasoline, which is buoyant in seawater, may be used to provide buoyancy. However, because gasoline is more compressible than water, its buoyancy decreases as the dive gets deeper. Therefore it is necessary to carry and drop ballast to achieve positive buoyancy for the return trip to the surface.

The most structurally efficient hull shape for airships and submarines has a circular cross-section. The buoyancy force passes through the center of the circle. Therefore, for roll stability the CG must be located below the hull centerline. Thus the crew compartment of an airship is placed beneath the hull to lower the CG.

3.7 Fluids in Rigid-Body Motion (on the Web)

3.8 Summary and Useful Equations

In this chapter we have reviewed the basic concepts of fluid statics. This included:

- ✓ Deriving the basic equation of fluid statics in vector form.
- ✓ Applying this equation to compute the pressure variation in a static fluid:
  - ○ Incompressible liquids: pressure increases uniformly with depth.
  - ○ Gases: pressure decreases nonuniformly with elevation (dependent on other thermodynamic properties).
- ✓ Study of:
  - ○ Gage and absolute pressure.
  - ○ Use of manometers and barometers.
- ✓ Analysis of the fluid force magnitude and location on submerged:
  - ○ Plane surfaces.
  - ○ Curved surfaces.
- ✓ *Derivation and use of Archimedes' Principle of Buoyancy.
- ✓ *Analysis of rigid-body fluid motion (on the Web).

Note: Most of the Useful Equations in the table below have a number of constraints or limitations—*be sure to refer to their page numbers for details*!

Useful Equations

Hydrostatic pressure variation:	$\frac{dp}{dz} = -\rho g \equiv -\gamma$	**(3.6)**	Page 59
Hydrostatic pressure variation (incompressible fluid):	$p - p_0 = \Delta p = \rho g h$	**(3.7)**	Page 61
Hydrostatic pressure variation (several incompressible fluids):	$\Delta p = g \sum_i \rho_i h_i$	**(3.8)**	Page 65
Hydrostatic force on submerged plane (integral form):	$F_R = \int_A p\, dA$	**(3.10a)**	Page 70
Hydrostatic force on submerged plane:	$F_R = p_c A$	**(3.10b)**	Page 70
Location y' of hydrostatic force on submerged plane (integral):	$y' F_R = \int_A y p\, dA$	**(3.11a)**	Page 71
Location y' of hydrostatic force on submerged plane (algebraic):	$y' = y_c + \frac{\rho g \sin\theta\, I_{\hat{x}\hat{x}}}{F_R}$	**(3.11b)**	Page 71
Location y' of hydrostatic force on submerged plane (p_0 neglected):	$y' = y_c + \frac{I_{\hat{x}\hat{x}}}{A y_c}$	**(3.11c)**	Page 71
Location x' of hydrostatic force on submerged plane (integral):	$x' F_R = \int_A x\, p\, dA$	**(3.12a)**	Page 72
Location x' of hydrostatic force on submerged plane (algebraic):	$x' = x_c + \frac{\rho g \sin\theta\, I_{\hat{x}\hat{y}}}{F_R}$	**(3.12b)**	Page 72
Location x' of hydrostatic force on submerged plane (p_0 neglected):	$x' = x_c + \frac{I_{\hat{x}\hat{y}}}{A\, y_c}$	**(3.12c)**	Page 72
Horizontal and vertical hydrostatic forces on curved submerged surface:	$F_H = p_c A$ and $F_V = \rho g V\!\!\!/$	**(3.15)**	Page 77
Buoyancy force on submerged object:	$F_{\text{buoyancy}} = \rho g V\!\!\!/$	**(3.16)**	Page 80

We have now concluded our introduction to the fundamental concepts of fluid mechanics, and the basic concepts of fluid statics. In the next chapter we will begin our study of fluids in motion.

*These topics apply to sections that may be omitted without loss of continuity in the text material.

Case Study

The Falkirk Wheel

The Falkirk Wheel.

Hydrostatics, the study of fluids at rest, is an ancient discipline, so one might think there are no new or exciting applications still to be developed. The Falkirk wheel in Scotland is a dramatic demonstration that this is not the case; it is a novel replacement for a lock, a device for moving a boat from one water level to another. The wheel, which has a diameter of 35 m, consists of two sets of axe-shaped opposing arms (which take the shape of a Celtic-inspired, double-headed axe). Sitting in bearings in the ends of these arms are two water-filled caissons, or tanks, each with a capacity of 80,000 gal. The hydrostatics concept of Archimedes' principle, which we studied in this chapter, states that floating objects displace their own weight of water. Hence, the boat shown entering the lower caisson displaces water from the caisson weighing exactly the same as the boat itself. This means the entire wheel remains balanced at all times (both caissons always carry the same weight, whether containing boats or not), and so, despite its enormous mass, it rotates through 180° in less than four minutes while using very little power. The electric motors used for this use 22.5 kilowatts (kW) of power, so the energy used in four minutes is about 1.5 kilowatt-hours (kWh); even at current prices, this works out to be only a few cents worth of energy.

References

1. Burcher, R., and L. Rydill, *Concepts in Submarine Design*. Cambridge, UK: Cambridge University Press, 1994.

2. Marchaj, C. A., *Aero-Hydrodynamics of Sailing*, rev. ed. Camden, ME: International Marine Publishing, 1988.

Problems

3.1 Compressed nitrogen (140 lbm) is stored in a spherical tank of diameter $D = 2.5$ ft at a temperature of 77°F. What is the pressure inside the tank? If the maximum allowable stress in the tank is 30 ksi, find the minimum theoretical wall thickness of the tank.

Standard Atmosphere

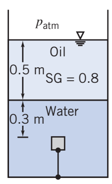
3.2 Because the pressure falls, water boils at a lower temperature with increasing altitude. Consequently, cake mixes and boiled eggs, among other foods, must be cooked different lengths of time. Determine the boiling temperature of water at 1000 and 2000 m elevation on a standard day, and compare with the sea-level value.

3.3 Ear "popping" is an unpleasant phenomenon sometimes experienced when a change in pressure occurs, for example in a fast-moving elevator or in an airplane. If you are in a two-seater airplane at 3000 m and a descent of 100 m causes your ears to "pop," what is the pressure change that your ears "pop" at, in millimeters of mercury? If the airplane now rises to 8000 m and again begins descending, how far will the airplane descend before your ears "pop" again? Assume a U.S. Standard Atmosphere.

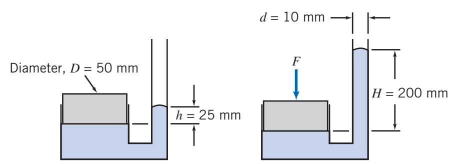
3.4 When you are on a mountain face and boil water, you notice that the water temperature is 195°F. What is your approximate altitude? The next day, you are at a location where it boils at 185°F. How high did you climb between the two days? Assume a U.S. Standard Atmosphere.

Pressure Variation in a Static Fluid

3.5 A 125-mL cube of solid oak is held submerged by a tether as shown. Calculate the actual force of the water on the bottom surface of the cube and the tension in the tether.



P3.5



P3.6

3.6 The tube shown is filled with mercury at 20°C. Calculate the force applied to the piston.

3.7 The following pressure and temperature measurements were taken by a meteorological balloon rising through the lower atmosphere:

p (psia)	14.71	14.62	14.53	14.45	14.36	14.27	14.18	14.1	14.01	13.92	13.84
T (°F)	53.6	52	50.9	50.4	50.2	50	50.5	51.4	52.9	54	53.8

The initial values (top of table) correspond to ground level. Using the ideal gas law ($p = \rho RT$ with $R = 53.3$ ft·lbf/lbm·°R), compute and plot the variation of air density (in lbm/ft^3) with height.

3.8 A hollow metal cube with sides 100 mm floats at the interface between a layer of water and a layer of SAE 10W oil such that 10% of the cube is exposed to the oil. What is the pressure difference between the upper and lower horizontal surfaces? What is the average density of the cube?

3.9 Your pressure gage indicates that the pressure in your cold tires is 0.25 MPa (gage) on a mountain at an elevation of 3500 m. What is the absolute pressure? After you drive down to sea level, your tires have warmed to 25°C. What pressure does your gage now indicate? Assume a U.S. Standard Atmosphere.

3.10 An air bubble, 0.3 in. in diameter, is released from the regulator of a scuba diver swimming 100 ft below the sea surface. (The water temperature is 86°F.) Estimate the diameter of the bubble just before it reaches the water surface.

3.11 A cube with 6 in. sides is suspended in a fluid by a wire. The top of the cube is horizontal and 8 in. below the free surface. If the cube has a mass of 2 slugs and the tension in the wire is $T = 50.7$ lbf, compute the fluid specific gravity, and from this determine the fluid. What are the gage pressures on the upper and lower surfaces?

3.12 Assuming the bulk modulus is constant for seawater, derive an expression for the density variation with depth, h, below the surface. Show that the result may be written

$$\rho \approx \rho_0 + bh$$

where ρ_0 is the density at the surface. Evaluate the constant b. Then, using the approximation, obtain an equation for the variation of pressure with depth below the surface. Determine the depth in feet at which the error in pressure predicted by the approximate solution is 0.01 percent.

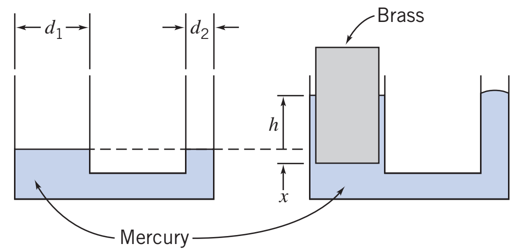
3.13 Oceanographic research vessels have descended to 6.5 mi below sea level. At these extreme depths, the compressibility of seawater can be significant. One may model the behavior of seawater by assuming that its bulk modulus remains constant. Using this assumption, evaluate the deviations in density and pressure compared with values computed using the incompressible assumption at a depth, h, of 6.5 mi in seawater. Express your answers as a percentage. Plot the results over the range $0 \le h \le 7$ mi.

3.14 An inverted cylindrical container is lowered slowly beneath the surface of a pool of water. Air trapped in the container is compressed isothermally as the hydrostatic pressure increases. Develop an expression for the water height, y, inside the container in terms of the container height, H, and depth of submersion, h. Plot y/H versus h/H.

3.15 You close the top of your straw with your thumb and lift the straw out of your glass containing Coke. Holding it vertically, the total length of the straw is 45 cm, but the Coke held in the straw is in the bottom 15 cm. What is the pressure in the straw just below your thumb? Ignore any surface tension effects.

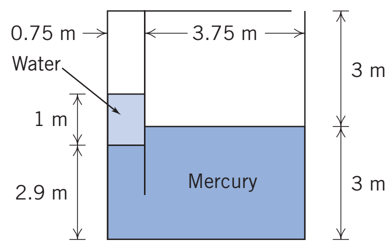
3.16 A water tank filled with water to a depth of 16 ft has in inspection cover (1 in. × 1 in.) at its base, held in place by a plastic bracket. The bracket can hold a load of 9 lbf. Is the bracket strong enough? If it is, what would the water depth have to be to cause the bracket to break?

3.17 A container with two circular vertical tubes of diameters $d_1 = 39.5$ mm and $d_2 = 12.7$ mm is partially filled with mercury. The equilibrium level of the liquid is shown in the left diagram. A cylindrical object made from solid brass is placed in the larger tube so that it floats, as shown in the right diagram. The object is $D = 37.5$ mm in diameter and $H = 76.2$ mm high. Calculate the pressure at the lower surface needed to float the object. Determine the new equilibrium level, h, of the mercury with the brass cylinder in place.



P3.17

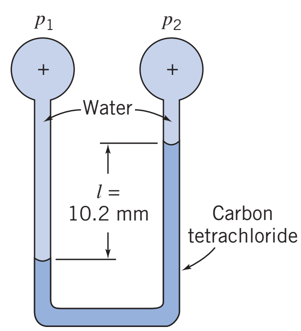
3.18 A partitioned tank as shown contains water and mercury. What is the gage pressure in the air trapped in the left chamber? What pressure would the air on the left need to be pumped to in order to bring the water and mercury free surfaces level?



P3.18, P3.19

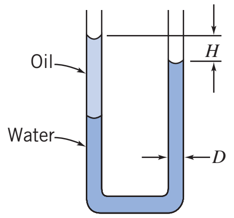
3.19 In the tank of Problem 3.18, if the opening to atmosphere on the right chamber is first sealed, what pressure would the air on the left now need to be pumped to in order to bring the water and mercury free surfaces level? (Assume the air trapped in the right chamber behaves isothermally.)

3.20 Consider the two-fluid manometer shown. Calculate the applied pressure difference.



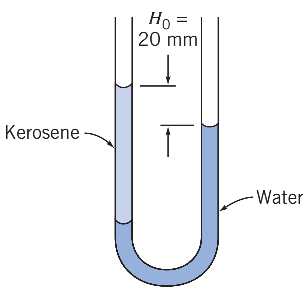
P3.20

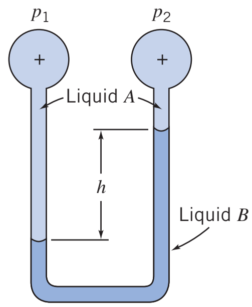
3.21 A manometer is formed from glass tubing with uniform inside diameter, $D = 6.35$ mm, as shown. The U-tube is partially filled with water. Then $\forall = 3.25\ cm^3$ of Meriam red oil is added to the left side. Calculate the equilibrium height, H, when both legs of the U-tube are open to the atmosphere.



P3.21

3.22 The manometer shown contains water and kerosene. With both tubes open to the atmosphere, the free-surface elevations differ by $H_0 = 20.0$ mm. Determine the elevation difference when a pressure of 98.0 Pa (gage) is applied to the right tube.

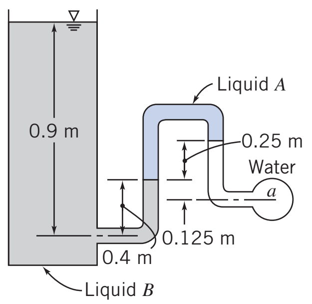




P3.22 P3.23

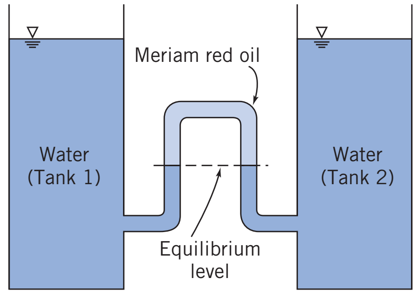
3.23 The manometer shown contains two liquids. Liquid A has SG = 0.88 and liquid B has SG = 2.95. Calculate the deflection, h, when the applied pressure difference is $p_1 - p_2 = 18\ lbf/ft^2$.

3.24 Determine the gage pressure in kPa at point a, if liquid A has SG = 1.20 and liquid B has SG = 0.75. The liquid surrounding point a is water, and the tank on the left is open to the atmosphere.



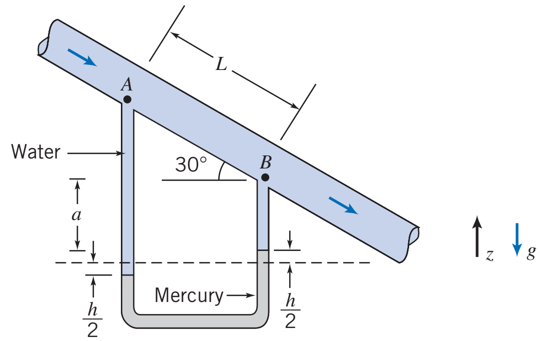
P3.24

3.25 An engineering research company is evaluating using a sophisticated \$80,000 laser system between two large water storage tanks. You suggest that the job can be done with a \$200 manometer arrangement. Oil less dense than water can be used to give a significant amplification of meniscus movement; a small difference in level between the tanks will cause a much larger deflection in the oil levels in the manometer. If you set up a rig using Meriam red oil as the manometer fluid, determine the amplification factor that will be seen in the rig.



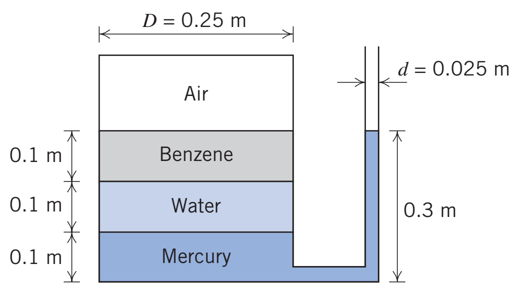
P3.25

3.26 Water flows downward along a pipe that is inclined at 30° below the horizontal, as shown. Pressure difference $p_A - p_B$ is due partly to gravity and partly to friction. Derive an algebraic expression for the pressure difference. Evaluate the pressure difference if $L = 5$ ft and $h = 6$ in.



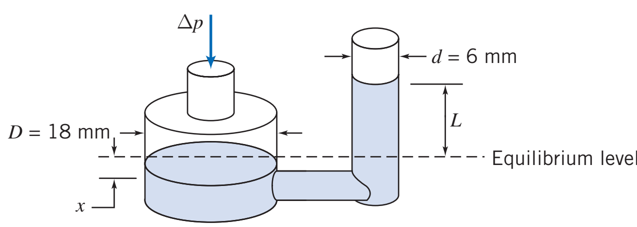
P3.26

3.27 Consider a tank containing mercury, water, benzene, and air as shown. Find the air pressure (gage). If an opening is made in the top of the tank, find the equilibrium level of the mercury in the manometer.



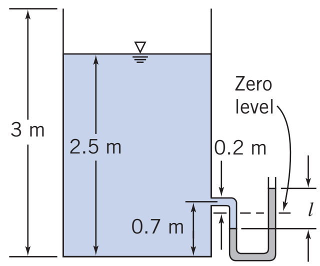
P3.27

3.28 A reservoir manometer has vertical tubes of diameter $D = 18$ mm and $d = 6$ mm. The manometer liquid is Meriam red oil. Develop an algebraic expression for liquid deflection L in the small tube when gage pressure Δp is applied to the reservoir. Evaluate the liquid deflection when the applied pressure is equivalent to 25 mm of water (gage).



P3.28

3.29 A rectangular tank, open to the atmosphere, is filled with water to a depth of 2.5 m as shown. A U-tube manometer is connected to the tank at a location 0.7 m above the tank bottom. If the zero level of the Meriam blue manometer fluid is 0.2 m below the connection, determine the deflection l after the manometer is connected and all air has been removed from the connecting leg.

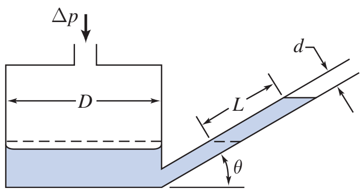


P3.29, P3.31, P3.37

3.30 A reservoir manometer is calibrated for use with a liquid of specific gravity 0.827. The reservoir diameter is 5/8 in. and the (vertical) tube diameter is 3/16 in. Calculate the required distance between marks on the vertical scale for 1 in. of water pressure difference.

3.31 The manometer fluid of Problem 3.29 is replaced with mercury (same zero level). The tank is sealed and the air pressure is increased to a gage pressure of 0.5 atm. Determine the deflection l.

3.32 The inclined-tube manometer shown has $D = 96$ mm and $d = 8$ mm. Determine the angle, θ, required to provide a 5 : 1 increase in liquid deflection, L, compared with the total deflection in a regular U-tube manometer. Evaluate the sensitivity of this inclined-tube manometer.

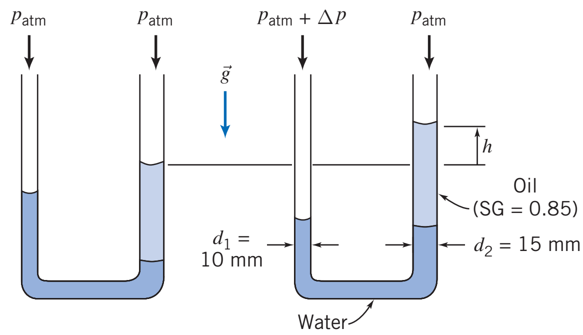


P3.32, P3.33

3.33 The inclined-tube manometer shown has $D = 76$ mm and $d = 8$ mm, and is filled with Meriam red oil. Compute the angle, θ, that will give a 15-cm oil deflection along the inclined tube for an applied pressure of 25 mm of water (gage). Determine the sensitivity of this manometer.

3.34 A barometer accidentally contains 6.5 inches of water on top of the mercury column (so there is also water vapor instead of a vacuum at the top of the barometer). On a day when the temperature is 70°F, the mercury column height is 28.35 inches (corrected for thermal expansion). Determine the barometric pressure in psia. If the ambient temperature increased to 85°F and the barometric pressure did not change, would the mercury column be longer, be shorter, or remain the same length? Justify your answer.

3.35 A student wishes to design a manometer with better sensitivity than a water-filled U-tube of constant diameter. The student's concept involves using tubes with different diameters and two liquids, as shown. Evaluate the deflection h of this manometer, if the applied pressure difference is $\Delta p = 250$ N/m^2. Determine the sensitivity of this manometer. Plot the manometer sensitivity as a function of the diameter ratio d_2/d_1.



P3.35

3.36 A water column stands 50 mm high in a 2.5-mm diameter glass tube. What would be the column height if the surface tension were zero? What would be the column height in a 1.0-mm diameter tube?

3.37 If the tank of Problem 3.29 is sealed tightly and water drains slowly from the bottom of the tank, determine the deflection, l, after the system has attained equilibrium.

3.38 Consider a small-diameter open-ended tube inserted at the interface between two immiscible fluids of different densities. Derive an expression for the height difference Δh between the interface level inside and outside the tube in terms of tube diameter D, the two fluid densities ρ_1 and ρ_2, and the surface tension σ and angle θ for the two fluids' interface. If the two fluids are water and mercury, find the height difference if the tube diameter is 40 mils (1 mil = 0.001 in.).

3.39 You have a manometer consisting of a tube that is 0.5 in. inner diameter (ID). On one side, the manometer leg contains mercury, 0.6 in.^{3} of an oil (SG = 1.4), and 0.2 in.^{3} of air as a bubble in the oil. The other leg contains only mercury. Both legs are open to the atmosphere and are in a static condition. An accident occurs in which 0.2 in.^{3} of the oil and the air bubble are removed from one leg. How much do the mercury height levels change?

3.40 Compare the height due to capillary action of water exposed to air in a circular tube of diameter $D = 0.5$ mm, and between two infinite vertical parallel plates of gap $a = 0.5$ mm.

3.41 Two vertical glass plates 12 in. × 12 in. are placed in an open tank containing water. At one end the gap between the plates is 0.004 in., and at the other it is 0.080 in. Plot the curve of water height between the plates from one end of the pair to the other.

3.42 Based on the atmospheric temperature data of the U.S. Standard Atmosphere of Fig. 3.3, compute and plot the pressure variation with altitude, and compare with the pressure data of Table A.3.

3.43 On a certain calm day, a mild inversion causes the atmospheric temperature to remain constant at 30°C between sea level and 5000-m altitude. Under these conditions, (a) calculate the elevation change for which a 3 percent reduction in air pressure occurs, (b) determine the change of elevation necessary to effect a 5 percent reduction in density, and (c) plot p_2/p_1 and ρ_2/ρ_1 as a function of Δz.

3.44 At ground level in Denver, Colorado, the atmospheric pressure and temperature are 83.2 kPa and 25°C. Calculate the pressure on Pike's Peak at an elevation of 2690 m above the city assuming (a) an incompressible and (b) an adiabatic atmosphere. Plot the ratio of pressure to ground level pressure in Denver as a function of elevation for both cases.

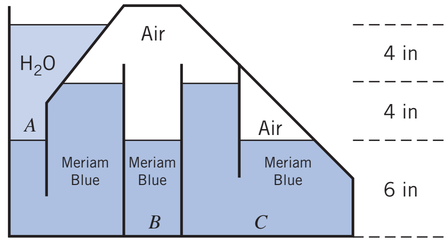
3.45 The Martian atmosphere behaves as an ideal gas with mean molecular mass of 32.0 and constant temperature of 200 K. The atmospheric density at the planet surface is $\rho = 0.015$ kg/m^3 and Martian gravity is 3.92 m/s^2. Calculate the density of the Martian atmosphere at height $z = 20$ km above the surface. Plot the ratio of density to surface density as a function of elevation. Compare with that for data on the Earth's atmosphere.

3.46 A door 1 m wide and 1.5 m high is located in a plane vertical wall of a water tank. The door is hinged along its upper edge, which is 1 m below the water surface. Atmospheric pressure acts on the outer surface of the door and at the water surface. (a) Determine the magnitude and line of action of the total resultant force from all fluids acting on the door. (b) If the water surface gage pressure is raised to 0.3 atm, what is the resultant force and where is its line of action? (c) Plot the ratios F/F_0 and y'/y_c for different values of the surface pressure ratio p_s/p_{atm}. (F_0 is the resultant force when $p_s = p_{atm}$.)

3.47 A door 1 m wide and 1.5 m high is located in a plane vertical wall of a water tank. The door is hinged along its upper edge, which is 1 m below the water surface. Atmospheric pressure acts on the outer surface of the door. (a) If the pressure at the water surface is atmospheric, what force must be applied at the lower edge of the door in order to keep the door from opening? (b) If the water surface gage pressure is raised to 0.5 atm, what force must be applied at the lower edge of the door to keep the door from opening? (c) Find the ratio F/F_0 as a function of the surface pressure ratio p_s/p_{atm}. (F_0 is the force required when $p_s = p_{atm}$.)

3.48 A hydropneumatic elevator consists of a piston-cylinder assembly to lift the elevator cab. Hydraulic oil, stored in an accumulator tank pressurized by air, is valved to the piston as needed to lift the elevator. When the elevator descends, oil is returned to the accumulator. Design the least expensive accumulator that can satisfy the system requirements. Assume the lift is 3 floors, the maximum load is 10 passengers, and the maximum system pressure is 800 kPa (gage). For column bending strength, the piston diameter must be at least 150 mm. The elevator cab and piston have a combined mass of 3000 kg, and are to be purchased. Perform the analysis needed to define, as a function of system operating pressure, the piston diameter, the accumulator volume and diameter, and the wall thickness. Discuss safety features that your company should specify for the complete elevator system. Would it be preferable to use a completely pneumatic design or a completely hydraulic design? Why?

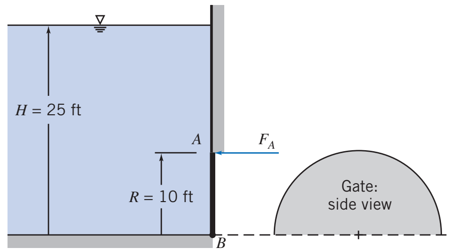
3.49 Find the pressures at points A, B, and C, as shown in the figure, and in the two air cavities.



P3.49

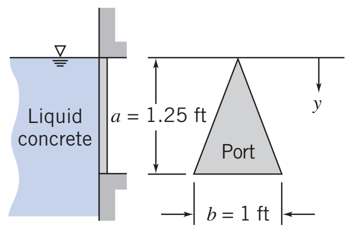
Hydrostatic Force on Submerged Surfaces

3.50 Semicircular plane gate AB is hinged along B and held by horizontal force F_A applied at A. The liquid to the left of the gate is water. Calculate the force F_A required for equilibrium.



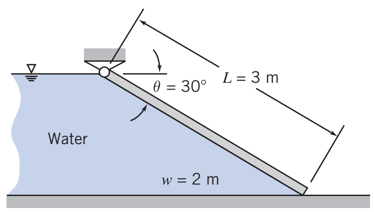
P3.50

3.51 A triangular access port must be provided in the side of a form containing liquid concrete. Using the coordinates and dimensions shown, determine the resultant force that acts on the port and its point of application.



P3.51

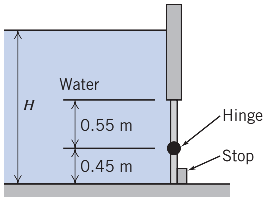
3.52 A plane gate of uniform thickness holds back a depth of water as shown. Find the minimum weight needed to keep the gate closed.



P3.52

3.53 Consider a semicylindrical trough of radius R and length L. Develop general expressions for the magnitude and line of action of the hydrostatic force on one end, if the trough is partially filled with water and open to atmosphere. Plot the results (in nondimensional form) over the range of water depth $0 \le d/R \le 1$.

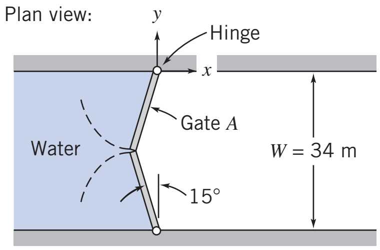
3.54 A rectangular gate (width $w = 2$ m) is hinged as shown, with a stop on the lower edge. At what depth H will the gate tip?



P3.54

3.55 For a mug of tea (65 mm diameter), imagine it cut symmetrically in half by a vertical plane. Find the force that each half experiences due to an 80-mm depth of tea.

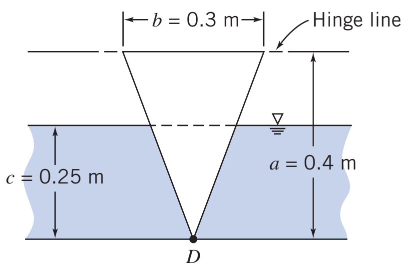
3.56 Gates in the Poe Lock at Sault Ste. Marie, Michigan, close a channel $W = 34$ m wide, $L = 360$ m long, and $D = 10$ m deep. The geometry of one pair of gates is shown; each gate is hinged at the channel wall. When closed, the gate edges are forced together at the center of the channel by water pressure. Evaluate the force exerted by the water on gate A. Determine the magnitude and direction of the force components exerted by the gate on the hinge. (Neglect the weight of the gate.)



P3.56

3.57 A section of vertical wall is to be constructed from ready-mix concrete poured between forms. The wall is to be 3 m high, 0.25 m thick, and 5 m wide. Calculate the force exerted by the ready-mix concrete on each form. Determine the line of application of the force.

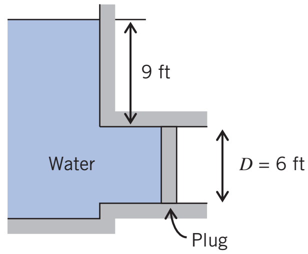
3.58 A window in the shape of an isosceles triangle and hinged at the top is placed in the vertical wall of a form that contains liquid concrete. Determine the minimum force that must be applied at point D to keep the window closed for the configuration of form and concrete shown. Plot the results over the range of concrete depth $0 \leq c \leq a$



P3.58

3.59 Solve Example 3.6 again using the two separate pressures method. Consider the distributed force to be the sum of a force F_1 caused by the uniform gage pressure and a force F_2 caused by the liquid. Solve for these forces and their lines of action. Then sum moments about the hinge axis to calculate F_t.

3.60 A large open tank contains water and is connected to a 6-ft-diameter conduit as shown. A circular plug is used to seal the conduit. Determine the magnitude, direction, and location of the force of the water on the plug.

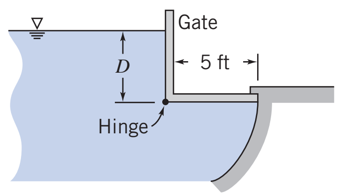


P3.60

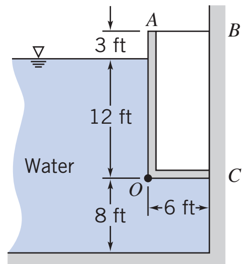
3.61 What holds up a car on its rubber tires? Most people would tell you that it is the air pressure inside the tires. However, the air pressure is the same all around the hub (inner wheel), and the air pressure inside the tire therefore pushes down from the top as much as it pushes up from below, having no net effect on the hub. Resolve this paradox by explaining where the force is that keeps the car off the ground.

3.62 The circular access port in the side of a water standpipe has a diameter of 0.6 m and is held in place by eight bolts evenly spaced around the circumference. If the standpipe diameter is 7 m and the center of the port is located 12 m below the free surface of the water, determine (a) the total force on the port and (b) the appropriate bolt diameter.

3.63 As water rises on the left side of the rectangular gate, the gate will open automatically. At what depth above the hinge will this occur? Neglect the mass of the gate.



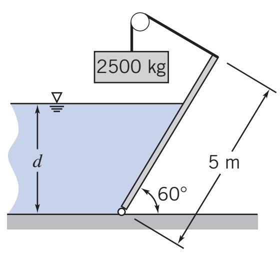
P3.63



P3.64

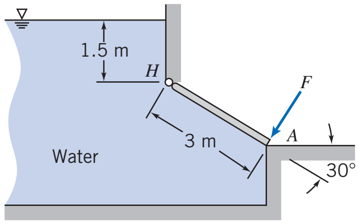
3.64 The gate AOC shown is 6 ft wide and is hinged along O. Neglecting the weight of the gate, determine the force in bar AB. The gate is sealed at C.

3.65 The gate shown is 3 m wide and for analysis can be considered massless. For what depth of water will this rectangular gate be in equilibrium as shown?



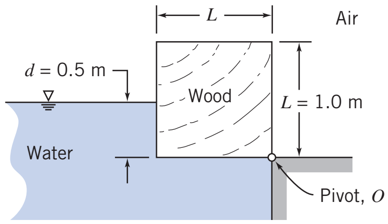
P3.65

3.66 The gate shown is hinged at H. The gate is 3 m wide normal to the plane of the diagram. Calculate the force required at A to hold the gate closed.



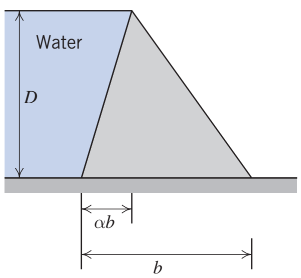
P3.66

3.67 A long, square wooden block is pivoted along one edge. The block is in equilibrium when immersed in water to the depth shown. Evaluate the specific gravity of the wood, if friction in the pivot is negligible.



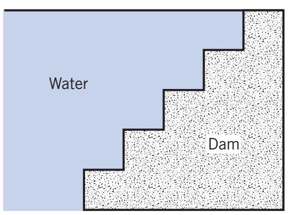
P3.67

3.68 A solid concrete dam is to be built to hold back a depth D of water. For ease of construction the walls of the dam must be planar. Your supervisor asks you to consider the following dam cross-sections: a rectangle, a right triangle with the hypotenuse in contact with the water, and a right triangle with the vertical in contact with the water. She wishes you to determine which of these would require the least amount of concrete. What will your report say? You decide to look at one more possibility: a nonright triangle, as shown. Develop and plot an expression for the cross-section area A as a function of a, and find the minimum cross-sectional area.



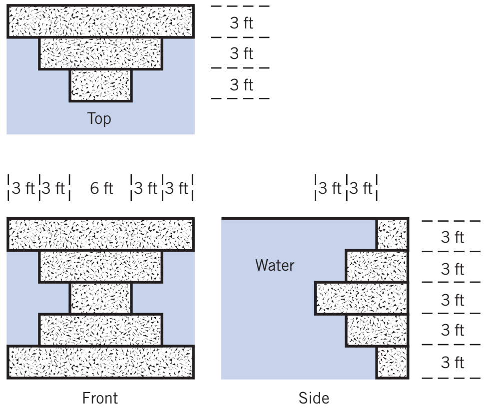
P3.68

3.69 For the geometry shown, what is the vertical force on the dam? The steps are 0.5 m high, 0.5 m deep, and 3 m wide.



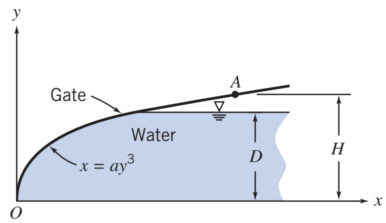
P3.69

3.70 For the dam shown, what is the vertical force of the water on the dam?



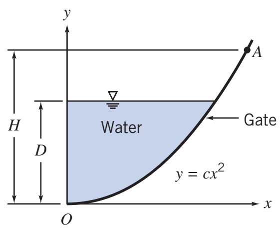
P3.70

3.71 The gate shown is 1.5 m wide and pivoted at O; $a = 1.0\ \text{m}^{-2}$, $D = 1.20$ m, and $H = 1.40$ m. Determine (a) the magnitude and moment of the vertical component of the force about O, and (b) the horizontal force that must be applied at point A to hold the gate in position.



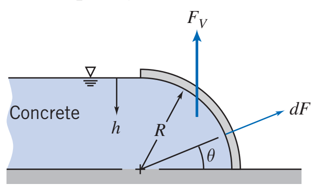
P3.71

3.72 The parabolic gate shown is 2 m wide and pivoted at O; $c = 0.25\ \text{m}^{-1}$, $D = 2$ m, and $H = 3$ m. Determine (a) the magnitude and line of action of the vertical force on the gate due to the water, (b) the horizontal force applied at A required to maintain the gate in equilibrium, and (c) the vertical force applied at A required to maintain the gate in equilibrium.



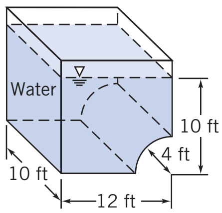
P3.72

3.73 Liquid concrete is poured into the form (R = 2 ft). The form is w = 15 ft wide normal to the diagram. Compute the magnitude of the vertical force exerted on the form by the concrete, and specify its line of action.



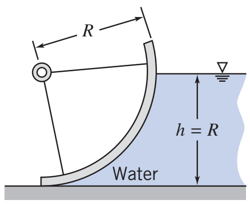
P3.73

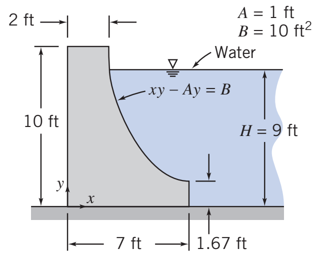
3.74 An open tank is filled with water to the depth indicated. Atmospheric pressure acts on all outer surfaces of the tank. Determine the magnitude and line of action of the vertical component of the force of the water on the curved part of the tank bottom.



P3.74

3.75 A spillway gate formed in the shape of a circular arc is w m wide. Find the magnitude and line of action of the vertical component of the force due to all fluids acting on the gate.

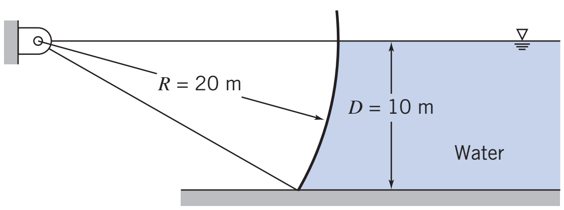




P3.75 P3.76

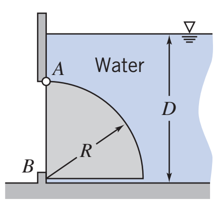
3.76 A dam is to be constructed using the cross-section shown. Assume the dam width is $w = 160$ ft. For water height $H = 9$ ft, calculate the magnitude and line of action of the vertical force of water on the dam face. Is it possible for water forces to overturn this dam? Under what circumstances will this happen?

3.77 A Tainter gate used to control water flow from the Uniontown Dam on the Ohio River is shown; the gate width is $w = 35$ m. Determine the magnitude, direction, and line of action of the force from the water acting on the gate.



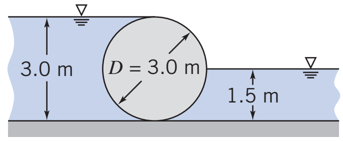
P3.77

3.78 A gate, in the shape of a quarter-cylinder, hinged at A and sealed at B, is 3 m wide. The bottom of the gate is 4.5 m below the water surface. Determine the force on the stop at B if the gate is made of concrete; $R = 3$ m.



P3.78

3.79 Consider the cylindrical weir of diameter 3 m and length 6 m. If the fluid on the left has a specific gravity of 1.6, and on the right has a specific gravity of 0.8, find the magnitude and direction of the resultant force.

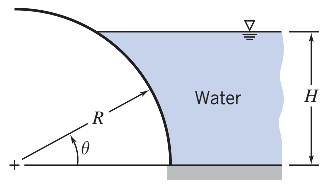


P3.79, P3.80

3.80 A cylindrical weir has a diameter of 3 m and a length of 6 m. Find the magnitude and direction of the resultant force acting on the weir from the water.

3.81 A cylindrical log of diameter D rests against the top of a dam. The water is level with the top of the log and the center of the log is level with the top of the dam. Obtain expressions for (a) the mass of the log per unit length and (b) the contact force per unit length between the log and dam.

3.82 A curved surface is formed as a quarter of a circular cylinder with $R = 0.750$ m as shown. The surface is $w = 3.55$ m wide. Water stands to the right of the curved surface to depth $H = 0.650$ m. Calculate the vertical hydrostatic force on the curved surface. Evaluate the line of action of this force. Find the magnitude and line of action of the horizontal force on the surface.

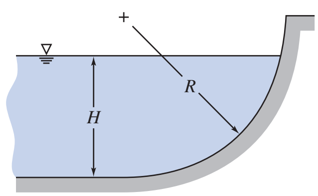


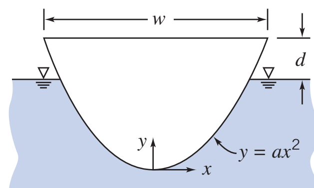
P3.82

Buoyancy and Stability

3.83 If you throw an anchor out of your canoe but the rope is too short for the anchor to rest on the bottom of the pond, will your canoe float higher, lower, or stay the same? Prove your answer.

3.84 A curved submerged surface, in the shape of a quarter cylinder with radius $R = 1.0$ ft is shown. The form can withstand a maximum vertical load of 350 lbf before breaking. The width is $w = 4$ ft. Find the maximum depth H to which the form may be filled. Find the line of action of the vertical force for this condition. Plot the results over the range of concrete depth $0 \leq H \leq R$.



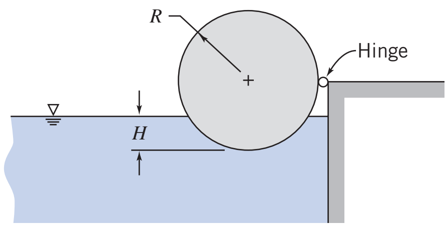


P3.84 P3.85

3.85 The cross-sectional shape of a canoe is modeled by the curve $y = ax^2$, where $a = 1.2\ \text{ft}^{-1}$ and the coordinates are in

feet. Assume the width of the canoe is constant at $w = 2$ ft over its entire length $L = 18$ ft. Set up a general algebraic expression relating the total mass of the canoe and its contents to distance d between the water surface and the gunwale of the floating canoe. Calculate the maximum total mass allowable without swamping the canoe.

3.86 The cylinder shown is supported by an incompressible liquid of density ρ, and is hinged along its length. The cylinder, of mass M, length L, and radius R, is immersed in liquid to depth H. Obtain a general expression for the cylinder specific gravity versus the ratio of liquid depth to cylinder radius, $\alpha = H/R$, needed to hold the cylinder in equilibrium for $0 \leq \alpha < 1$. Plot the results.

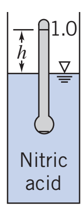


P3.86

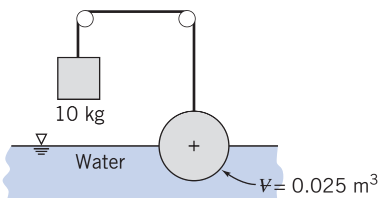
3.87 A canoe is represented by a right semicircular cylinder, with $R = 1.2$ ft and $L = 17$ ft. The canoe floats in water that is $d = 1$ ft deep. Set up a general algebraic expression for the total mass (canoe and contents) that can be floated, as a function of depth. Evaluate for the given conditions. Plot the results over the range of water depth $0 \leq d \leq R$.

3.88 A glass observation room is to be installed at the corner of the bottom of an aquarium. The aquarium is filled with seawater to a depth of 35 ft. The glass is a segment of a sphere, radius 5 ft, mounted symmetrically in the corner. Compute the magnitude and direction of the net force on the glass structure.

***3.89** A hydrometer is a specific gravity indicator, the value being indicated by the level at which the free surface intersects the stem when floating in a liquid. The 1.0 mark is the level when in distilled water. For the unit shown, the immersed volume in distilled water is 15 cm^3. The stem is 6 mm in diameter. Find the distance, h, from the 1.0 mark to the surface when the hydrometer is placed in a nitric acid solution of specific gravity 1.5.



P3.89



P3.90

***3.90** Find the specific weight of the sphere shown if its volume is 0.025m^3. State all assumptions. What is the equilibrium position of the sphere if the weight is removed?

***3.91** The fat-to-muscle ratio of a person may be determined from a specific gravity measurement. The measurement is made by immersing the body in a tank of water and measuring the net weight. Develop an expression for the specific gravity of a person in terms of their weight in air, net weight in water, and SG $= f(T)$ for water.

***3.92** Quantify the statement, "Only the tip of an iceberg shows (in seawater)."

***3.93** An open tank is filled to the top with water. A steel cylindrical container, wall thickness $\delta = 1$ mm, outside diameter $D = 100$ mm, and height $H = 1$ m, with an open top, is gently placed in the water. What is the volume of water that overflows from the tank? How many 1 kg weights must be placed in the container to make it sink? Neglect surface tension effects.

***3.94** Quantify the experiment performed by Archimedes to identify the material content of King Hiero's crown. Assume you can measure the weight of the king's crown in air, W_a, and the weight in water, W_w. Express the specific gravity of the crown as a function of these measured values.

***3.95** Gas bubbles are released from the regulator of a submerged scuba diver. What happens to the bubbles as they rise through the seawater? Explain.

***3.96** Hot-air ballooning is a popular sport. According to a recent article, "hot-air volumes must be large because air heated to 150°F over ambient lifts only 0.018 lbf/ft^3 compared to 0.066 and 0.071 for helium and hydrogen, respectively." Check these statements for sea-level conditions. Calculate the effect of increasing the hot-air maximum temperature to 250°F above ambient.

***3.97** Hydrogen bubbles are used to visualize water flow streaklines in the video, *Flow Visualization*. A typical hydrogen bubble diameter is $d = 0.001$ in. The bubbles tend to rise slowly in water because of buoyancy; eventually they reach terminal speed relative to the water. The drag force of the water on a bubble is given by $F_D = 3\pi\mu V d$, where μ is the viscosity of water and V is the bubble speed relative to the water. Find the buoyancy force that acts on a hydrogen bubble immersed in water. Estimate the terminal speed of a bubble rising in water.

***3.98** It is desired to use a hot air balloon with a volume of 320,000 ft^3 for rides planned in summer morning hours when the air temperature is about 48°F. The torch will warm the air inside the balloon to a temperature of 160°F. Both inside and outside pressures will be "standard" (14.7 psia). How much mass can be carried by the balloon (basket, fuel, passengers, personal items, and the component of the balloon itself) if neutral buoyancy is to be assured? What mass can be carried by the balloon to ensure vertical takeoff acceleration of 2.5 ft/s^2? For this, consider that both balloon and inside air have to be accelerated, as well as some of the surrounding air (to make way for the balloon). The rule of thumb is that the total mass subject to acceleration is the mass of the balloon, all its appurtenances, and twice its volume of air. Given that the volume of hot air is fixed during the flight, what can the balloonists do when they want to go down?

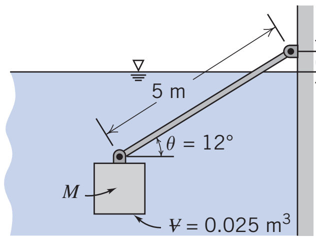
***3.99** Scientific balloons operating at pressure equilibrium with the surroundings have been used to lift instrument packages to extremely high altitudes. One such balloon,

*These problems require material from sections that may be omitted without loss of continuity in the text material.

filled with helium, constructed of polyester with a skin thickness of 0.013 mm and a diameter of 120 m, lifted a payload of 230 kg. The specific gravity of the skin material is 1.28. Determine the altitude to which the balloon would rise. Assume that the helium used in the balloon is in thermal equilibrium with the ambient air, and that the balloon is a perfect sphere.

***3.100** A helium balloon is to lift a payload to an altitude of 40 km, where the atmospheric pressure and temperature are 3.0 mbar and −25°C, respectively. The balloon skin is polyester with specific gravity of 1.28 and thickness of 0.015 mm. To maintain a spherical shape, the balloon is pressurized to a gage pressure of 0.45 mbar. Determine the maximum balloon diameter if the allowable tensile stress in the skin is limited to 62 MN/m^2. What payload can be carried?

***3.101** A block of volume 0.025 m^3 is allowed to sink in water as shown. A circular rod 5 m long and 20 cm^2 in cross-section is attached to the weight and also to the wall. If the rod mass is 1.25 kg and the rod makes an angle of 12 degrees with the horizontal at equilibrium, what is the mass of the block?



P3.101

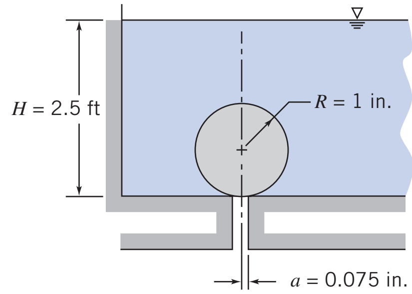
***3.102** The stem of a glass hydrometer used to measure specific gravity is 5 mm in diameter. The distance between marks on the stem is 2 mm per 0.1 increment of specific gravity. Calculate the magnitude and direction of the error introduced by surface tension if the hydrometer floats in kerosene. (Assume the contact angle between kerosene and glass is 0°.)

***3.103** A sphere, of radius R, is partially immersed, to depth d, in a liquid of specific gravity SG. Obtain an algebraic expression for the buoyancy force acting on the sphere as a function of submersion depth d. Plot the results over the range of water depth $0 \leq d \leq 2R$.

***3.104** If the mass M in Problem 3.101 is released from the rod, at equilibrium how much of the rod will remain submerged? What will be the minimum required upward force at the tip of the rod to just lift it out of the water?

***3.105** In a logging operation, timber floats downstream to a lumber mill. It is a dry year, and the river is running low, as low as 60 cm in some locations. What is the largest diameter log that may be transported in this fashion (leaving a minimum 5 cm clearance between the log and the bottom of the river)? For the wood, SG = 0.8.

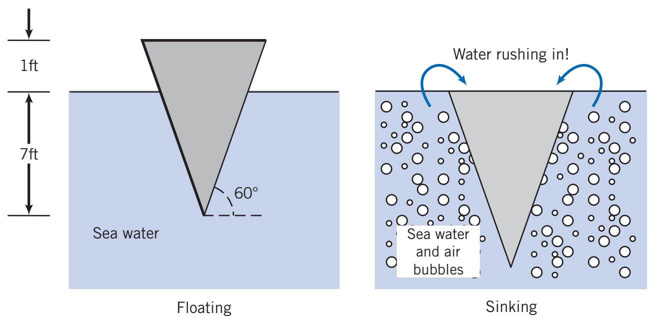
***3.106** A sphere of radius 1 in., made from material of specific gravity of SG = 0.95, is submerged in a tank of water. The sphere is placed over a hole of radius 0.075 in., in the tank bottom. When the sphere is released, will it stay on the bottom of the tank or float to the surface?



P3.106

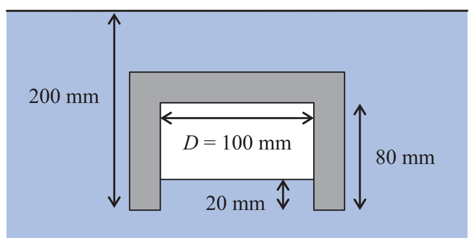
***3.107** A cylindrical timber, with $D = 1$ ft and $L = 15$ ft, is weighted on its lower end so that it floats vertically with 10 ft submerged in seawater. When displaced vertically from its equilibrium position, the timber oscillates or "heaves" in a vertical direction upon release. Estimate the frequency of oscillation in this heave mode. Neglect viscous effects and water motion.

***3.108** You are in the Bermuda Triangle when you see a bubble plume eruption (a large mass of air bubbles, similar to a foam) off to the side of the boat. Do you want to head toward it and be part of the action? What is the effective density of the water and air bubbles in the drawing on the right that will cause the boat to sink? Your boat is 10 ft long, and weight is the same in both cases.



P3.108

***3.109** A bowl is inverted symmetrically and held in a dense fluid, SG = 15.6, to a depth of 200 mm measured along the centerline of the bowl from the bowl rim. The bowl height is 80 mm, and the fluid rises 20 mm inside the bowl. The bowl is 100 mm inside diameter, and it is made from an old clay recipe, SG = 6.1. The volume of the bowl itself is about 0.9 L. What is the force required to hold it in place?



P3.109

*These problems require material from sections that may be omitted without loss of continuity in the text material.

***3.110** In the "Cartesian diver" child's toy, a miniature "diver" is immersed in a column of liquid. When a diaphragm at the top of the column is pushed down, the diver sinks to the bottom. When the diaphragm is released, the diver again rises. Explain how the toy might work.

***3.111** Consider a conical funnel held upside down and submerged slowly in a container of water. Discuss the force needed to submerge the funnel if the spout is open to the atmosphere. Compare with the force needed to submerge the funnel when the spout opening is blocked by a rubber stopper.

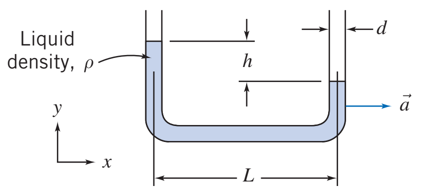
***3.112** Three steel balls (each about half an inch in diameter) lie at the bottom of a plastic shell floating on the water surface in a partially filled bucket. Someone removes the steel balls from the shell and carefully lets them fall to the bottom of the bucket, leaving the plastic shell to float empty. What happens to the water level in the bucket? Does it rise, go down, or remain unchanged? Explain.

***3.113** A proposed ocean salvage scheme involves pumping air into "bags" placed within and around a wrecked vessel on the sea bottom. Comment on the practicality of this plan, supporting your conclusions with analyses.

Fluids in Rigid-Body Motion

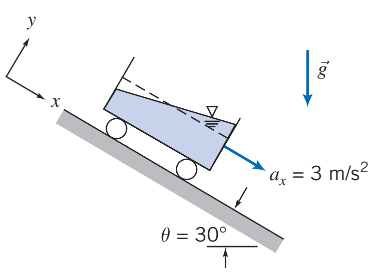
***3.114** A cylindrical container, similar to that analyzed in Example 3.10 (on the Web), is rotated at a constant rate of 2 Hz about its axis. The cylinder is 0.5 m in diameter and initially contains water that is 0.3 m deep. Determine the height of the liquid free surface at the center of the container. Does your answer depend on the density of the liquid? Explain.

***3.115** A crude accelerometer can be made from a liquid-filled U-tube as shown. Derive an expression for the liquid level difference h caused by an acceleration $\vec{a}$, in terms of the tube geometry and fluid properties.



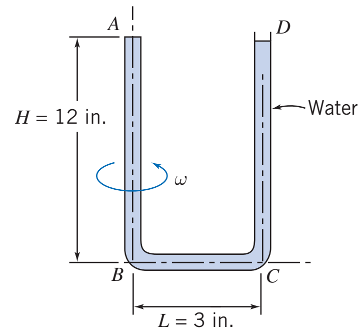
P3.115

***3.116** A rectangular container of water undergoes constant acceleration down an incline as shown. Determine the slope of the free surface using the coordinate system shown.



P3.116

***3.117** The U-tube shown is filled with water at $T = 68°$F. It is sealed at A and open to the atmosphere at D. The tube is rotated about vertical axis AB at 1600 rpm. For the dimensions shown, would cavitation occur in the tube?



P3.117, P3.118

***3.118** If the U-tube of Problem 3.117 is spun at 300 rpm, what will the pressure be at A? If a small leak appears at A, how much water will be lost at D?

***3.119** A centrifugal micromanometer can be used to create small and accurate differential pressures in air for precise measurement work. The device consists of a pair of parallel disks that rotate to develop a radial pressure difference. There is no flow between the disks. Obtain an expression for pressure difference in terms of rotation speed, radius, and air density. Evaluate the speed of rotation required to develop a differential pressure of 8 μm of water using a device with a 50 mm radius.

***3.120** A test tube is spun in a centrifuge. The tube support is mounted on a pivot so that the tube swings outward as rotation speed increases. At high speeds, the tube is nearly horizontal. Find (a) an expression for the radial component of acceleration of a liquid element located at radius r, (b) the radial pressure gradient dp/dr, and (c) the required angular velocity to generate a pressure of 250 MPa in the bottom of a test tube containing water. (The free surface and bottom radii are 50 and 130 mm, respectively.)

***3.121** A rectangular container, of base dimensions 0.4 m × 0.2 m and height 0.4 m, is filled with water to a depth of 0.2 m; the mass of the empty container is 10 kg. The container is placed on a plane inclined at 30° to the horizontal. If the coefficient of sliding friction between the container and the plane is 0.3, determine the angle of the water surface relative to the horizontal.

***3.122** If the container of Problem 3.121 slides without friction, determine the angle of the water surface relative to the horizontal. What is the slope of the free surface for the same acceleration up the plane?

***3.123** A cubical box, 80 cm on a side, half-filled with oil (SG = 0.80), is given a constant horizontal acceleration of 0.25 g parallel to one edge. Determine the slope of the free surface and the pressure along the horizontal bottom of the box.

***3.124** Gas centrifuges are used in one process to produce enriched uranium for nuclear fuel rods. The maximum

*These problems require material from sections that may be omitted without loss of continuity in the text material.

peripheral speed of a gas centrifuge is limited by stress considerations to about 950 ft/s. Assume a gas centrifuge containing uranium hexafluoride gas, with molecular gas $M_m = 352$, and ideal gas behavior. Develop an expression for the ratio of maximum pressure to pressure at the centrifuge axis. Evaluate the pressure ratio for a gas temperature of 620°F.

***3.125** A pail, 400 mm in diameter and 400 mm deep, weighs 15 N and contains 200 mm of water. The pail is swung in a vertical circle of 1-m radius at a speed of 5 m/s. Assume the water moves as a rigid body. At the instant when the pail is at the top of its trajectory, compute the tension in the string and the pressure on the bottom of the pail from the water.

***3.126** A partially full can of soda is placed at the outer edge of a child's merry-go-round, located $R = 5$ ft from the axis of rotation. The can diameter and height are 2.5 in. and 5 in., respectively. The can is half full, and the soda has specific gravity SG = 1.05. Evaluate the slope of the liquid surface in the can if the merry-go-round spins at 20 rpm. Calculate the spin rate at which the can would spill, assuming no slippage between the can bottom and the merry-go-round. Would the can most likely spill or slide off the merry-go-round?

***3.127** When a water polo ball is submerged below the surface in a swimming pool and released from rest, it is observed to pop out of the water. How would you expect the height to which it rises above the water to vary with depth of submersion below the surface? Would you expect the same results for a beach ball? For a table-tennis ball?

***3.128** Cast iron or steel molds are used in a horizontal-spindle machine to make tubular castings such as liners and tubes. A charge of molten metal is poured into the spinning mold. The radial acceleration permits nearly uniformly thick wall sections to form. A steel liner, of length $L = 6$ ft, outer radius $r_o = 6$ in., and inner radius $r_i = 4$ in., is to be formed by this process. To attain nearly uniform thickness, the angular velocity should be at least 300 rpm. Determine (a) the resulting radial acceleration on the inside surface of the liner and (b) the maximum and minimum pressures on the surface of the mold.

***3.129** The analysis of Problem 3.121 suggests that it may be possible to determine the coefficient of sliding friction between two surfaces by measuring the slope of the free surface in a liquid-filled container sliding down an inclined surface. Investigate the feasibility of this idea.

*These problems require material from sections that may be omitted without loss of continuity in the text material.

4

Basic Equations in Integral Form for a Control Volume

Case Study in Energy and the Environment

Wave Power: *Pelamis Wave Energy Converter*

As we have seen in earlier Case Studies in Energy and the Environment, there is a lot of renewable energy in ocean waves that could be exploited. A good example of a machine for doing this is the *Pelamis Wave Energy Converter* developed by *Pelamis Wave Power Ltd*. in Scotland. This machine was the world's first commercial-scale machine to generate power and supply it to the power grid from offshore wave energy, and the first to be used in a commercial wave farm project.

Schematic of possible *Pelamis* wave farm. (Picture courtesy of Pelamis Wave Power Ltd.)

The wave-powered electrical generating machine consists of a partially submerged, articulated structure made up of cylindrical sections connected by hinged joints. As waves pass over the structure, the flexing motion of the joints (generated by buoyancy forces, discussed in Chapter 3) is resisted by an arrangement of hydraulic "rams" inside the cylindrical sections; these rams are then used to pump high-pressure fluid through hydraulic motors, which ultimately drive electrical generators to produce electricity. The power that is generated in each joint is sent down a single cable to a junction device on the sea bed; several devices can be connected together (as suggested in the schematic) and linked to shore through a single seabed cable.

The latest generation of machines are 180 meters long (they have four sections, each 45 meters long) and 4 meters in diameter, with four power conversion modules. Each machine can generate up to 750 kilowatts, depending on the specific environmental conditions at the site; they will produce 25–40 percent of the full rated output, on average, over the course of a year. Hence each machine can provide sufficient power to meet the annual electricity demand of about 500 homes. This is not a future technology; three first-generation machines have already been installed off the coast of Portugal, and a single machine is being built and a four-unit machine (generating 3 megawatts of power) is planned for use off the northern coast of Scotland. *Pelamis Wave Power Ltd*. has also expressed interest in installing Pelamis machines off the coast of Cornwall in England, and in the Pacific Ocean off the coast of Tillamook, Oregon. The *Pelamis* machine has a number of advantages: It is durable and low maintenance, uses available technology, and generates electricity inexpensively.

We are now ready to study fluids in motion, so we have to decide how we are to examine a flowing fluid. There are two options available to us, discussed in Chapter 1:

1. We can study the motion of an *individual fluid particle or group of particles* as they move through space. This is the *system* approach, which has the advantage that the physical laws (e.g., Newton's second law, $\vec{F} = d\vec{P}/dt$, where $\vec{F}$ is the force and $d\vec{P}/dt$ is the rate of momentum change of the fluid) apply to matter and hence directly to the system. One disadvantage is that in practice the math associated with this approach can become somewhat complicated, usually leading to a set of partial differential equations. We will look at this approach in detail in Chapter 5. The system approach is needed if we are interested in studying the trajectory of particles over time, for example, in pollution studies.
2. We can study a *region of space* as fluid flows through it, which is the *control volume* approach. This is very often the method of choice, because it has widespread practical application; for example, in aerodynamics we are usually interested in the lift and drag on a wing (which we select as part of the control volume) rather than what happens to individual fluid particles. The disadvantage of this approach is that

the physical laws apply to matter and not directly to regions of space, so we have to perform some math to convert physical laws from their system formulation to a control volume formulation.

We will examine the control volume approach in this chapter. The alert reader will notice that this chapter has the word *integral* in its title, and Chapter 5 has the word *differential*. This is an important distinction: It indicates that we will study a finite region in this chapter and the motion of a particle (an infinitesimal) in Chapter 5 (although in Section 4.4 we will look at a differential control volume to derive the famous Bernoulli equation). The agenda for this chapter is to review the physical laws as they apply to a system (Section 4.1); develop some math to convert from a system to a control volume (Section 4.2) description; and obtain formulas for the physical laws for control volume analysis by combining the results of Sections 4.1 and 4.2.

4.1 Basic Laws for a System

The basic laws we will apply are conservation of mass, Newton's second law, the angular-momentum principle, and the first and second laws of thermodynamics. For converting these system equations to equivalent control volume formulas, it turns out we want to express each of the laws as a rate equation.

Conservation of Mass

For a system (by definition a fixed amount of matter, M, we have chosen) we have the simple result that $M = \text{constant}$. However, as discussed above, we wish to express each physical law as a rate equation, so we write

$$\left.\frac{dM}{dt}\right)_{\text{system}} = 0 \tag{4.1a}$$

where

$$M_{\text{system}} = \int_{M(\text{system})} dm = \int_{\mathcal{V}(\text{system})} \rho \, d\mathcal{V} \tag{4.1b}$$

Newton's Second Law

For a system moving relative to an inertial reference frame, Newton's second law states that the sum of all external forces acting on the system is equal to the time rate of change of linear momentum of the system,

$$\vec{F} = \left.\frac{d\vec{P}}{dt}\right)_{\text{system}} \tag{4.2a}$$

where the linear momentum of the system is given by

$$\vec{P}_{\text{system}} = \int_{M(\text{system})} \vec{V} \, dm = \int_{\mathcal{V}(\text{system})} \vec{V} \rho \, d\mathcal{V} \tag{4.2b}$$

The Angular-Momentum Principle

The angular-momentum principle for a system states that the rate of change of angular momentum is equal to the sum of all torques acting on the system,

$$\vec{T} = \left.\frac{d\vec{H}}{dt}\right)_{\text{system}} \tag{4.3a}$$

where the angular momentum of the system is given by

$$\vec{H}_{\text{system}} = \int_{M(\text{system})} \vec{r} \times \vec{V}\, dm = \int_{V\!\!\!/(\text{system})} \vec{r} \times \vec{V}\, \rho\, dV\!\!\!/ \tag{4.3b}$$

Torque can be produced by surface and body forces (here gravity) and also by shafts that cross the system boundary,

$$\vec{T} = \vec{r} \times \vec{F}_s + \int_{M(\text{system})} \vec{r} \times \vec{g}\, dm + \vec{T}_{\text{shaft}} \tag{4.3c}$$

The First Law of Thermodynamics

The first law of thermodynamics is a statement of conservation of energy for a system,

$$\delta Q - \delta W = dE$$

The equation can be written in rate form as

$$\dot{Q} - \dot{W} = \left.\frac{dE}{dt}\right)_{\text{system}} \tag{4.4a}$$

where the total energy of the system is given by

$$E_{\text{system}} = \int_{M(\text{system})} e\, dm = \int_{V\!\!\!/(\text{system})} e\, \rho\, dV\!\!\!/ \tag{4.4b}$$

and

$$e = u + \frac{V^2}{2} + gz \tag{4.4c}$$

In Eq. 4.4a, $\dot{Q}$ (the rate of heat transfer) is positive when heat is added to the system from the surroundings; $\dot{W}$ (the rate of work) is positive when work is done by the system on its surroundings. In Eq. 4.4c, u is the specific internal energy, V the speed, and z the height (relative to a convenient datum) of a particle of substance having mass dm.

The Second Law of Thermodynamics

If an amount of heat, δQ, is transferred to a system at temperature T, the second law of thermodynamics states that the change in entropy, dS, of the system satisfies

$$dS \geq \frac{\delta Q}{T}$$

On a rate basis we can write

$$\left.\frac{dS}{dt}\right)_{\text{system}} \geq \frac{1}{T}\dot{Q} \tag{4.5a}$$

where the total entropy of the system is given by

$$S_{\text{system}} = \int_{M(\text{system})} s\, dm = \int_{\forall(\text{system})} s\, \rho\, d\forall \tag{4.5b}$$

4.2 Relation of System Derivatives to the Control Volume Formulation

We now have the five basic laws expressed as system rate equations. Our task in this section is to develop a general expression for converting a system rate equation into an equivalent control volume equation. Instead of converting the equations for rates of change of M, $\vec{P}$, $\vec{H}$, E, and S (Eqs. 4.1a, 4.2a, 4.3a, 4.4a, and 4.5a) one by one, we let all of them be represented by the symbol N. Hence N represents the amount of mass, or momentum, or angular momentum, or energy, or entropy of the system. Corresponding to this extensive property, we will also need the intensive (i.e., per unit mass) property η. Thus

$$N_{\text{system}} = \int_{M(\text{system})} \eta\, dm = \int_{\forall(\text{system})} \eta\, \rho\, d\forall \tag{4.6}$$

Comparing Eq. 4.6 with Eqs. 4.1b, 4.2b, 4.3b, 4.4b, and 4.5b, we see that if:

$$\begin{aligned}
N &= M, && \text{then } \eta = 1\\
N &= \vec{P}, && \text{then } \eta = \vec{V}\\
N &= \vec{H}, && \text{then } \eta = \vec{r}\times\vec{V}\\
N &= E, && \text{then } \eta = e\\
N &= S, && \text{then } \eta = s
\end{aligned}$$

How can we derive a control volume description from a system description of a fluid flow? Before specifically answering this question, we can describe the derivation in general terms. We imagine selecting an arbitrary piece of the flowing fluid at some time t_0, as shown in Fig. 4.1*a*—we could imagine dyeing this piece of fluid, say, blue.

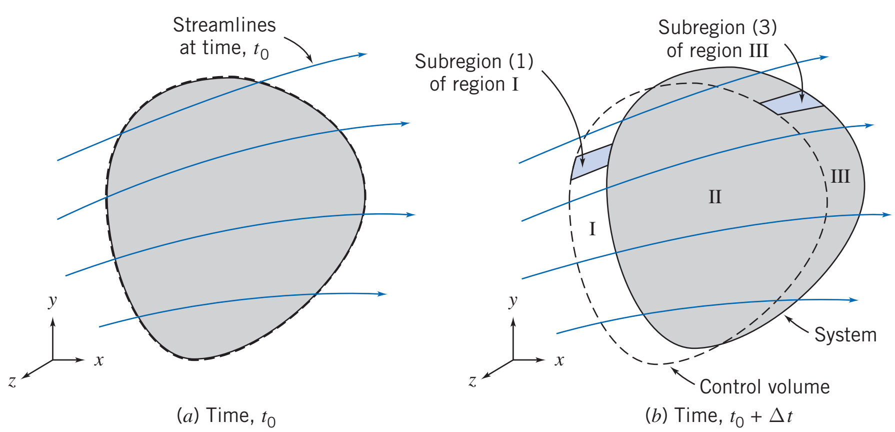


Fig. 4.1 System and control volume configuration.

This initial shape of the fluid system is chosen as our control volume, which is fixed in space relative to coordinates xyz. After an infinitesimal time Δt the system will have moved (probably changing shape as it does so) to a new location, as shown in Fig. 4.1*b*. The laws we discussed above apply to this piece of fluid—for example, its mass will be constant (Eq. 4.1a). By examining the geometry of the system/control volume pair at $t = t_0$ and at $t = t_0 + \Delta t$, we will be able to obtain control volume formulations of the basic laws.

Derivation

From Fig. 4.1 we see that the system, which was entirely within the control volume at time t_0, is partially out of the control volume at time $t_0 + \Delta t$. In fact, three regions can be identified. These are: regions I and II, which together make up the control volume, and region III, which, with region II, is the location of the system at time $t_0 + \Delta t$.

Recall that our objective is to relate the rate of change of any arbitrary extensive property, N, of the system to quantities associated with the control volume. From the definition of a derivative, the rate of change of N_{system} is given by

$$\left.\frac{dN}{dt}\right)_{\text{system}} \equiv \lim_{\Delta t \to 0} \frac{N_s)_{t_0+\Delta t} - N_s)_{t_0}}{\Delta t} \tag{4.7}$$

For convenience, subscript s has been used to denote the system in the definition of a derivative in Eq. 4.7.

From the geometry of Fig. 4.1,

$$N_s)_{t_0+\Delta t} = (N_{\text{II}} + N_{\text{III}})_{t_0+\Delta t} = (N_{\text{CV}} - N_{\text{I}} + N_{\text{III}})_{t_0+\Delta t}$$

and

$$N_s)_{t_0} = (N_{\text{CV}})_{t_0}$$

Substituting into the definition of the system derivative, Eq. 4.7, we obtain

$$\left.\frac{dN}{dt}\right)_s = \lim_{\Delta t \to 0} \frac{(N_{\text{CV}} - N_{\text{I}} + N_{\text{III}})_{t_0+\Delta t} - N_{\text{CV}})_{t_0}}{\Delta t}$$

Since the limit of a sum is equal to the sum of the limits, we can write

$$\left.\frac{dN}{dt}\right)_s = \underbrace{\lim_{\Delta t \to 0} \frac{N_{\text{CV}})_{t_0+\Delta t} - N_{\text{CV}})_{t_0}}{\Delta t}}_{①} + \underbrace{\lim_{\Delta t \to 0} \frac{N_{\text{III}})_{t_0+\Delta t}}{\Delta t}}_{②} - \underbrace{\lim_{\Delta t \to 0} \frac{N_{\text{I}})_{t_0+\Delta t}}{\Delta t}}_{③} \tag{4.8}$$

Our task now is to evaluate each of the three terms in Eq. 4.8.

Term ① in Eq. 4.8 simplifies to

$$\lim_{\Delta t \to 0} \frac{N_{\text{CV}})_{t_0+\Delta t} - N_{\text{CV}})_{t_0}}{\Delta t} = \frac{\partial N_{\text{CV}}}{\partial t} = \frac{\partial}{\partial t}\int_{\text{CV}} \eta\,\rho\, d\forall \tag{4.9a}$$

To evaluate term ② we first develop an expression for $N_{\text{III}})_{t_0+\Delta t}$ by looking at the enlarged view of a typical subregion (subregion (3)) of region III shown in Fig. 4.2. The vector area element $d\vec{A}$ of the control surface has magnitude dA, and its direction is the *outward* normal of the area element. In general, the velocity vector $\vec{V}$ will be at some angle α with respect to $d\vec{A}$.

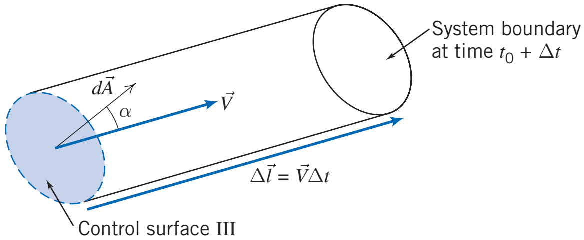


Fig. 4.2 Enlarged view of subregion (3) from Fig. 4.1.

For this subregion we have

$$dN_{\text{III}})_{t_0+\Delta t} = (\eta\, \rho\, d\mathcal{V})_{t_0+\Delta t}$$

We need to obtain an expression for the volume $d\mathcal{V}$ of this cylindrical element. The vector length of the cylinder is given by $\Delta\vec{l} = \vec{V}\Delta t$. The volume of a prismatic cylinder, whose area $d\vec{A}$ is at an angle α to its length $\Delta\vec{l}$, is given by $d\mathcal{V} = \Delta l\, dA \cos\alpha = \Delta\vec{l} \cdot d\vec{A} = \vec{V} \cdot d\vec{A}\Delta t$. Hence, for subregion (3) we can write

$$dN_{\text{III}})_{t_0+\Delta t} = \eta\, \rho \vec{V} \cdot d\vec{A}\Delta t$$

Then, for the entire region III we can integrate and for term 2 in Eq. 4.8 obtain

$$\lim_{\Delta t \to 0} \frac{N_{\text{III}})_{t_0+\Delta t}}{\Delta t} = \lim_{\Delta t \to 0} \frac{\int_{\text{CS}_{\text{III}}} dN_{\text{III}})_{t_0+\Delta t}}{\Delta t} = \lim_{\Delta t \to 0} \frac{\int_{\text{CS}_{\text{III}}} \eta\rho\vec{V} \cdot d\vec{A}\Delta t}{\Delta t} = \int_{\text{CS}_{\text{III}}} \eta\rho\vec{V} \cdot d\vec{A} \tag{4.9b}$$

We can perform a similar analysis for subregion (1) of region I, and obtain for term in Eq. 4.8

$$\lim_{\Delta t \to 0} \frac{N_{\text{I}})_{t_0+\Delta t}}{\Delta t} = -\int_{\text{CS}_{\text{I}}} \eta\, \rho\vec{V} \cdot d\vec{A} \tag{4.9c}$$

For subregion (1), the velocity vector acts *into* the control volume, but the area normal *always* (by convention) points outward (angle $\alpha > \pi/2$), so the scalar product in Eq. 4.9c is negative. Hence the minus sign in Eq. 4.9c is needed to cancel the negative result of the scalar product to make sure we obtain a positive result for the amount of matter that was in region I (we can't have negative matter).

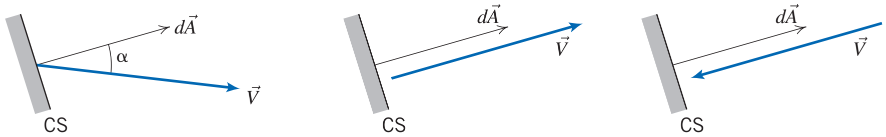
This concept of the sign of the scalar product is illustrated in Fig. 4.3 for (a) the general case of an inlet or exit, (b) an exit velocity parallel to the surface normal, and (c) an inlet velocity parallel to the surface normal. Cases (b) and (c) are obviously convenient special cases of (a); the value of the cosine in case (a) automatically generates the correct sign of either an inlet or an exit.

We can finally use Eqs. 4.9a, 4.9b, and 4.9c in Eq. 4.8 to obtain

$$\left.\frac{dN}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} \eta\, \rho\, d\mathcal{V} + \int_{\text{CS}_{\text{I}}} \eta\, \rho\vec{V} \cdot d\vec{A} + \int_{\text{CS}_{\text{III}}} \eta\, \rho\vec{V} \cdot d\vec{A}$$

and the two last integrals can be combined because CS_{I} and CS_{III} constitute the entire control surface,

$$\left.\frac{dN}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} \eta\, \rho\, d\mathcal{V} + \int_{\text{CS}} \eta\, \rho\vec{V} \cdot d\vec{A} \tag{4.10}$$



$\vec{V} \cdot d\vec{A} = VdA \cos \alpha$ (a) General inlet/exit

$\vec{V} \cdot d\vec{A} = +VdA$ (b) Normal exit

$\vec{V} \cdot d\vec{A} = -VdA$ (c) Normal inlet

Fig. 4.3 Evaluating the scalar product.

Equation 4.10 is the relation we set out to obtain. It is the fundamental relation between the rate of change of any arbitrary extensive property, N, of a system and the variations of this property associated with a control volume. Some authors refer to Eq. 4.10 as the *Reynolds Transport Theorem*.

Physical Interpretation

It took several pages, but we have reached our goal: We now have a formula (Eq. 4.10) that we can use to convert the rate of change of any extensive property N of a system to an equivalent formulation for use with a control volume. We can now use Eq. 4.10 in the various basic physical law equations (Eqs. 4.1a, 4.2a, 4.3a, 4.4a, and 4.5a) one by one, with N replaced with each of the properties M, $\vec{P}$, $\vec{H}$, E, and S (with corresponding symbols for η), to replace system derivatives with control volume expressions. Because we consider the equation itself to be "basic" we repeat it to emphasize its importance:

$$\left.\frac{dN}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} \eta\,\rho\, d\forall + \int_{\text{CS}} \eta\,\rho \vec{V} \cdot d\vec{A} \tag{4.10}$$

We need to be clear here: The system is the matter that happens to be passing through the chosen control volume, at the instant we chose. For example, if we chose as a control volume the region contained by an airplane wing and an imaginary rectangular boundary around it, the system would be the mass of the air that is instantaneously contained between the rectangle and the airfoil. Before applying Eq. 4.10 to the physical laws, let's discuss the meaning of each term of the equation:

$\left.\frac{dN}{dt}\right)_{\text{system}}$ is the rate of change of the system extensive property N. For example, if $N = \vec{P}$, we obtain the rate of change of momentum.

$\frac{\partial}{\partial t}\int_{\text{CV}} \eta\,\rho\, d\forall$ is the rate of change of the amount of property N in the control volume. The term $\int_{\text{CV}} \eta\,\rho\, d\forall$ computes the instantaneous value of N in the control volume ($\int_{\text{CV}} \rho\, d\forall$ is the instantaneous mass in the control volume). For example, if $N = \vec{P}$, then $\eta = \vec{V}$ and $\int_{\text{CV}} \vec{V}\rho d\forall$ computes the instantaneous amount of momentum in the control volume.

$\int_{\text{CS}} \eta\,\rho \vec{V} \cdot d\vec{A}$ is the rate at which property N is exiting the surface of the control volume. The term $\rho\vec{V} \cdot d\vec{A}$ computes the rate of mass transfer leaving across control surface area element $d\vec{A}$; multiplying by η computes the rate of flux of property N across the element; and integrating therefore computes the net flux of N out of the control volume. For example, if $N = \vec{P}$, then $\eta = \vec{V}$ and $\int_{\text{CS}} \vec{V}\rho\vec{V} \cdot d\vec{A}$ computes the net flux of momentum out of the control volume.

We make two comments about velocity $\vec{V}$ in Eq. 4.10. First, we reiterate the discussion for Fig. 4.3 that care should be taken in evaluating the dot product: Because $\vec{A}$ is always directed outwards, the dot product will be positive when $\vec{V}$ is outward and

negative when $\vec{V}$ is inward. Second, $\vec{V}$ is measured with respect to the control volume: When the control volume coordinates xyz are stationary or moving with a constant linear velocity, the control volume will constitute an inertial frame and the physical laws (specifically Newton's second law) we have described will apply.[1]

With these comments we are ready to combine the physical laws (Eqs. 4.1a, 4.2a, 4.3a, 4.4a, and 4.5a) with Eq. 4.10 to obtain some useful control volume equations.

4.3 Conservation of Mass

The first physical principle to which we apply this conversion from a system to a control volume description is the mass conservation principle: The mass of the system remains constant,

$$\left.\frac{dM}{dt}\right)_{\text{system}} = 0 \tag{4.1a}$$

where

$$M_{\text{system}} = \int_{M(\text{system})} dm = \int_{\forall(\text{system})} \rho\, d\forall \tag{4.1b}$$

The system and control volume formulations are related by Eq. 4.10,

$$\left.\frac{dN}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} \eta\,\rho\, d\forall + \int_{\text{CS}} \eta\,\rho \vec{V}\cdot d\vec{A} \tag{4.10}$$

where

$$N_{\text{system}} = \int_{M(\text{system})} \eta\, dm = \int_{\forall(\text{system})} \eta\,\rho\, d\forall \tag{4.6}$$

To derive the control volume formulation of conservation of mass, we set

$$N = M \qquad \text{and} \qquad \eta = 1$$

With this substitution, we obtain

$$\left.\frac{dM}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} \rho\, d\forall + \int_{\text{CS}} \rho \vec{V}\cdot d\vec{A} \tag{4.11}$$

Comparing Eqs. 4.1a and 4.11, we arrive (after rearranging) at the control volume formulation of the conservation of mass:

$$\frac{\partial}{\partial t}\int_{\text{CV}} \rho\, d\forall + \int_{\text{CS}} \rho \vec{V}\cdot d\vec{A} = 0 \tag{4.12}$$

In Eq. 4.12 the first term represents the rate of change of mass within the control volume; the second term represents the net rate of mass flux out through the control surface. Equation 4.12 indicates that the rate of change of mass in the control volume plus the net outflow is zero. The mass conservation equation is also called the

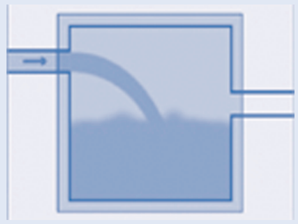
[1]For an accelerating control volume (one whose coordinates xyz are accelerating with respect to an "absolute" set of coordinates XYZ), we must modify the form of Newton's second law (Eq. 4.2a). We will do this in Sections 4.6 (linear acceleration) and 4.7 (arbitrary acceleration).

continuity equation. In common-sense terms, the rate of increase of mass in the control volume is due to the net inflow of mass:

$$\begin{array}{ccc} \text{Rate of increase} & = & \text{Net influx of} \\ \text{of mass in CV} & & \text{mass} \\ \dfrac{\partial}{\partial t}\displaystyle\int_{\mathrm{CV}} \rho\, d\mathcal{V} & = & -\displaystyle\int_{\mathrm{CS}} \rho \vec{V} \cdot d\vec{A} \end{array}$$

VIDEO

Mass Conservation: Filling a Tank.



Once again, we note that in using Eq. 4.12, care should be taken in evaluating the scalar product $\vec{V} \cdot d\vec{A} = VdA\cos\alpha$: It could be positive (outflow, $\alpha < \pi/2$), negative (inflow, $\alpha > \pi/2$), or even zero ($\alpha = \pi/2$). Recall that Fig. 4.3 illustrates the general case as well as the convenient cases $\alpha = 0$ and $\alpha = \pi$.

Special Cases

In special cases it is possible to simplify Eq. 4.12. Consider first the case of an incompressible fluid, in which density remains constant. When ρ is constant, it is not a function of space or time. Consequently, for *incompressible fluids*, Eq. 4.12 may be written as

$$\rho \frac{\partial}{\partial t}\int_{\mathrm{CV}} d\mathcal{V} + \rho \int_{\mathrm{CS}} \vec{V} \cdot d\vec{A} = 0$$

The integral of $d\mathcal{V}$ over the control volume is simply the volume of the control volume. Thus, on dividing through by ρ, we write

$$\frac{\partial \mathcal{V}}{\partial t} + \int_{\mathrm{CS}} \vec{V} \cdot d\vec{A} = 0$$

For a nondeformable control volume of fixed size and shape, $\mathcal{V}$ = constant. The conservation of mass for incompressible flow through a fixed control volume becomes

$$\int_{\mathrm{CS}} \vec{V} \cdot d\vec{A} = 0 \tag{4.13a}$$

A useful special case is when we have (or can approximate) uniform velocity at each inlet and exit. In this case Eq. 4.13a simplifies to

$$\sum_{\mathrm{CS}} \vec{V} \cdot \vec{A} = 0 \tag{4.13b}$$

Note that we have not assumed the flow to be steady in reducing Eq. 4.12 to the forms 4.13a and 4.13b. We have only imposed the restriction of incompressible fluid. Thus Eqs. 4.13a and 4.13b are statements of conservation of mass for flow of an incompressible fluid that may be steady or unsteady.

The dimensions of the integrand in Eq. 4.13a are L^3/t. The integral of $\vec{V} \cdot d\vec{A}$ over a section of the control surface is commonly called the *volume flow rate* or *volume rate of flow*. Thus, for incompressible flow, the volume flow rate into a fixed control volume must be equal to the volume flow rate out of the control volume. The volume flow rate Q, through a section of a control surface of area A, is given by

$$Q = \int_A \vec{V} \cdot d\vec{A} \tag{4.14a}$$

The average velocity magnitude, $\vec{V}$, at a section is defined as

$$\vec{V} = \frac{Q}{A} = \frac{1}{A}\int_A \vec{V}\cdot d\vec{A} \qquad (4.14b)$$

Consider now the general case of *steady, compressible flow* through a fixed control volume. Since the flow is steady, this means that at most $\rho = \rho(x, y, z)$. By definition, no fluid property varies with time in a steady flow. Consequently, the first term of Eq. 4.12 must be zero and, hence, for steady flow, the statement of conservation of mass reduces to

$$\int_{\text{CS}} \rho\vec{V}\cdot d\vec{A} = 0 \qquad (4.15a)$$

A useful special case is when we have (or can approximate) uniform velocity at each inlet and exit. In this case, Eq. 4.15a simplifies to

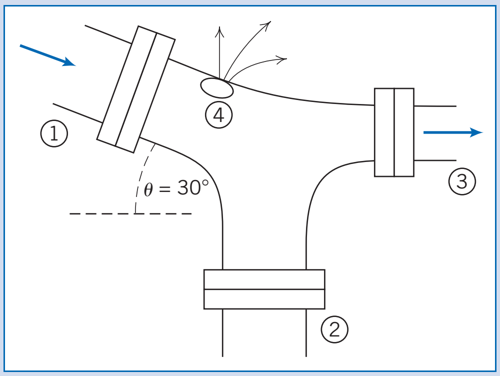
$$\sum_{\text{CS}} \rho\vec{V}\cdot\vec{A} = 0 \qquad (4.15b)$$

Thus, for steady flow, the mass flow rate into a control volume must be equal to the mass flow rate out of the control volume.

We will now look at three Examples to illustrate some features of the various forms of the conservation of mass equation for a control volume. Example 4.1 involves a problem in which we have uniform flow at each section, Example 4.2 involves a problem in which we do not have uniform flow at a location, and Example 4.3 involves a problem in which we have unsteady flow.

Example 4.1 MASS FLOW AT A PIPE JUNCTION

Consider the steady flow in a water pipe joint shown in the diagram. The areas are: $A_1 = 0.2$ m^2, $A_2 = 0.2$ m^2, and $A_3 = 0.15$ m^2. In addition, fluid is lost out of a hole at 4, estimated at a rate of 0.1 m^3/s. The average speeds at sections 1 and 3 are $V_1 = 5$ m/s and $V_3 = 12$ m/s, respectively. Find the velocity at section 2.



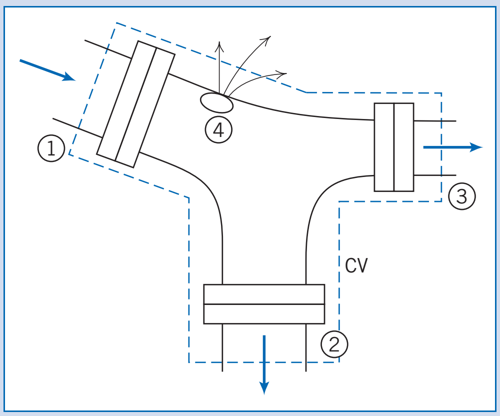
Given: Steady flow of water through the device.

$$A_1 = 0.2\,\text{m}^2 \quad A_2 = 0.2\,\text{m}^2 \quad A_3 = 0.15\,\text{m}^2$$
$$V_1 = 5\,\text{m/s} \quad V_3 = 12\,\text{m/s} \quad \rho = 999\,\text{kg/m}^3$$

Volume flow rate at 4 = 0.1 m^3/s

Find: Velocity at section 2.

Solution: Choose a fixed control volume as shown. Make an assumption that the flow at section 2 is outwards, and label the diagram accordingly (if this assumption is incorrect our final result will tell us).



Governing equation: The general control volume equation is Eq. 4.12, but we can go immediately to Eq. 4.13b because of assumptions (2) and (3) below,

$$\sum_{\text{CS}} \vec{V}\cdot\vec{A} = 0$$

Assumptions: (1) Steady flow (given).
(2) Incompressible flow.
(3) Uniform properties at each section.

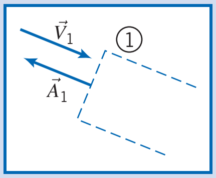
Hence (using Eq. 4.14a for the leak)

$$\vec{V}_1 \cdot \vec{A}_1 + \vec{V}_2 \cdot \vec{A}_2 + \vec{V}_3 \cdot \vec{A}_3 + Q_4 = 0 \tag{1}$$

where Q_4 is the flow rate out of the leak.

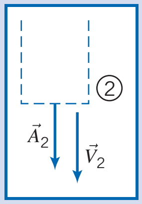
Let us examine the first three terms in Eq. 1 in light of the discussion of Fig. 4.3 and the directions of the velocity vectors:

$$\vec{V}_1 \cdot \vec{A}_1 = -V_1 A_1$$



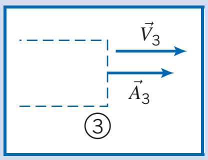
$\left\{ \begin{array}{l} \text{Sign of } \vec{V}_1 \cdot \vec{A}_1 \text{ is} \\ \text{negative at surface } 1 \end{array} \right\}$

$$\vec{V}_2 \cdot \vec{A}_2 = +V_2 A_2$$



$\left\{ \begin{array}{l} \text{Sign of } \vec{V}_2 \cdot \vec{A}_2 \text{ is} \\ \text{positive at surface } 2 \end{array} \right\}$

$$\vec{V}_3 \cdot \vec{A}_3 = +V_3 A_3$$



$\left\{ \begin{array}{l} \text{Sign of } \vec{V}_3 \cdot \vec{A}_3 \text{ is} \\ \text{positive at surface } 3 \end{array} \right\}$

Using these results in Eq. 1,

$$-V_1 A_1 + V_2 A_2 + V_3 A_3 + Q_4 = 0$$

or

$$V_2 = \frac{V_1 A_1 - V_3 A_3 - Q_4}{A_2}$$

$$= \frac{5\,\frac{\text{m}}{\text{s}} \times 0.2\,\text{m}^2 - 12\,\frac{\text{m}}{\text{s}} \times 0.15\,\text{m}^2 - \frac{0.1\,\text{m}^3}{\text{s}}}{0.2\,\text{m}^2}$$

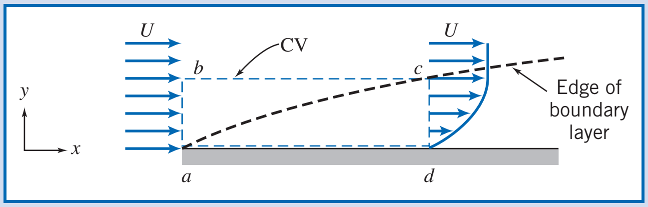
$$= -4.5\,\text{m/s} \longleftarrow V_2$$

Recall that V_2 represents the magnitude of the velocity, which we assumed was outwards from the control volume. The fact that V_2 is negative means that in fact we have an *inflow* at location 2—our initial assumption was invalid.

This problem demonstrates use of the sign convention for evaluating $\int_A \vec{V} \cdot d\vec{A}$ or $\Sigma_{CS} \vec{V} \cdot \vec{A}$. In particular, the area normal is *always* drawn *outwards* from the control surface.

Example 4.2 MASS FLOW RATE IN BOUNDARY LAYER

The fluid in direct contact with a stationary solid boundary has zero velocity; there is no slip at the boundary. Thus the flow over a flat plate adheres to the plate surface and forms a boundary layer, as depicted below. The flow ahead of the plate is uniform with velocity $\vec{V} = U\hat{i}$; $U = 30$ m/s. The velocity distribution within the boundary layer ($0 \le y \le \delta$) along cd is approximated as $u/U = 2(y/\delta) - (y/\delta)^2$.



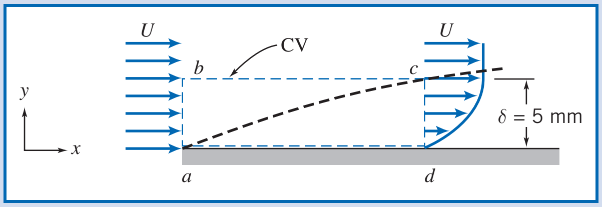
The boundary-layer thickness at location d is $\delta = 5$ mm. The fluid is air with density $\rho = 1.24$ kg/m^3. Assuming the plate width perpendicular to the paper to be $w = 0.6$ m, calculate the mass flow rate across surface bc of control volume $abcd$.

Given: Steady, incompressible flow over a flat plate, $\rho = 1.24$ kg/m^3. Width of plate, $w = 0.6$ m. Velocity ahead of plate is uniform: $\vec{V} = U\hat{i}$, $U = 30$ m/s.

At $x = x_d$:

$$\delta = 5\,\text{mm}$$

$$\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2$$



Find: Mass flow rate across surface bc.

Solution: The fixed control volume is shown by the dashed lines.

Governing equation: The general control volume equation is Eq. 4.12, but we can go immediately to Eq. 4.15a because of assumption (1) below,

$$\int_{CS} \rho \vec{V} \cdot d\vec{A} = 0$$

Assumptions:
(1) Steady flow (given).
(2) Incompressible flow (given).
(3) Two-dimensional flow, given properties are independent of z.

Assuming that there is no flow in the z direction, then

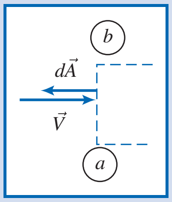
$$\int_{A_{ab}} \rho \vec{V} \cdot d\vec{A} + \int_{A_{bc}} \rho \vec{V} \cdot d\vec{A} + \int_{A_{cd}} \rho \vec{V} \cdot d\vec{A} + \underbrace{\cancel{\int_{A_{da}} \rho \vec{V} \cdot d\vec{A}}}_{\text{no flow across } da} = 0$$

$$\therefore \dot{m}_{bc} = \int_{A_{bc}} \rho \vec{V} \cdot d\vec{A} = -\int_{A_{ab}} \rho \vec{V} \cdot d\vec{A} - \int_{A_{cd}} \rho \vec{V} \cdot d\vec{A} \qquad (1)$$

We need to evaluate the integrals on the right side of the equation.

For depth w in the z direction, we obtain

$$\int_{A_{ab}} \rho \vec{V} \cdot d\vec{A} = -\int_{A_{ab}} \rho u \, dA = -\int_{y_a}^{y_b} \rho u w \, dy$$



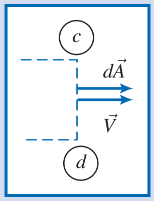
$$\left\{ \begin{array}{c} \vec{V} \cdot d\vec{A} \text{ is negative} \\ dA = w dy \end{array} \right\}$$

$$\{u = U \text{ over area } ab\}$$

$$= -\int_0^{\delta} \rho u w \, dy = -\int_0^{\delta} \rho U w \, dy$$

$$\int_{A_{ab}} \rho \vec{V} \cdot d\vec{A} = -\left[\rho U w y\right]_0^{\delta} = -\rho U w \delta$$

$$\int_{A_{cd}} \rho \vec{V} \cdot d\vec{A} = \int_{A_{cd}} \rho u \, dA = \int_{y_d}^{y_c} \rho u w \, dy$$



$$\left\{ \begin{array}{c} \vec{V} \cdot d\vec{A} \text{ is positive} \\ dA = w dy \end{array} \right\}$$

$$= \int_0^{\delta} \rho u w \, dy = \int_0^{\delta} \rho w U \left[2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2 \right] dy$$

$$\int_{A_{cd}} \rho \vec{V} \cdot d\vec{A} = \rho w U \left[\frac{y^2}{\delta} - \frac{y^3}{3\delta^2} \right]_0^{\delta} = \rho w U \delta \left[1 - \frac{1}{3} \right] = \frac{2\rho U w \delta}{3}$$

Substituting into Eq. 1, we obtain

$$\therefore \dot{m}_{bc} = \rho U w \delta - \frac{2\rho U w \delta}{3} = \frac{\rho U w \delta}{3}$$

$$= \frac{1}{3} \times 1.24 \frac{\text{kg}}{\text{m}^3} \times 30 \frac{\text{m}}{\text{s}} \times 0.6 \text{ m} \times 5 \text{ mm} \times \frac{\text{m}}{1000 \text{ mm}}$$

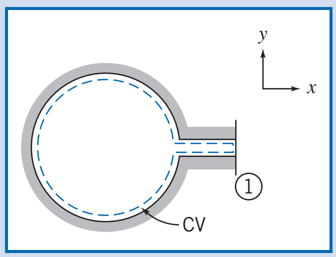
$$\left\{ \begin{array}{l} \text{Positive sign indicates flow} \\ \text{out across surface } bc. \end{array} \right\}$$

$$\dot{m}_{bc} = 0.0372 \text{ kg/s} \longleftarrow \dot{m}_b$$

This problem demonstrates use of the conservation of mass equation when we have nonuniform flow at a section.

Example 4.3 DENSITY CHANGE IN VENTING TANK

A tank of 0.05 m^3 volume contains air at 800 kPa (absolute) and 15°C. At $t = 0$, air begins escaping from the tank through a valve with a flow area of 65 mm^2. The air passing through the valve has a speed of 300 m/s and a density of 6 kg/m^3. Determine the instantaneous rate of change of density in the tank at $t = 0$.



Given: Tank of volume $V\!\!\!/ = 0.05 \text{ m}^3$ contains air at $p = 800$ kPa (absolute), $T = 15°\text{C}$. At $t = 0$, air escapes through a valve. Air leaves with speed $V = 300$ m/s and density $\rho = 6$ kg/m^3 through area $A = 65$ mm^2.

Find: Rate of change of air density in the tank at $t = 0$.

Solution: Choose a fixed control volume as shown by the dashed line.

Governing equation: $$\frac{\partial}{\partial t} \int_{\text{CV}} \rho \, dV\!\!\!/ + \int_{\text{CS}} \rho \vec{V} \cdot d\vec{A} = 0$$

Assumptions: (1) Properties in the tank are uniform, but time-dependent.
(2) Uniform flow at section 1.

Since properties are assumed uniform in the tank at any instant, we can take ρ out from within the volume integral of the first term,

$$\frac{\partial}{\partial t}\left[\rho_{\mathrm{CV}}\int_{\mathrm{CV}} d\forall\right] + \int_{\mathrm{CS}} \rho\vec{V}\cdot d\vec{A} = 0$$

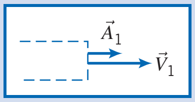
Now, $\int_{\mathrm{CV}} d\forall = \forall$, and hence

$$\frac{\partial}{\partial t}(\rho\forall)_{\mathrm{CV}} + \int_{\mathrm{CS}} \rho\vec{V}\cdot d\vec{A} = 0$$

The only place where mass crosses the boundary of the control volume is at surface 1. Hence

$$\int_{\mathrm{CS}} \rho\vec{V}\cdot d\vec{A} = \int_{A_1} \rho\vec{V}\cdot d\vec{A} \qquad \text{and} \qquad \frac{\partial}{\partial t}(\rho\forall) + \int_{A_1} \rho\vec{V}\cdot d\vec{A} = 0$$

At surface 1 the sign of $\rho\vec{V}\cdot d\vec{A}$ is positive, so



$$\frac{\partial}{\partial t}(\rho\forall) + \int_{A_1} \rho V\, dA = 0$$

Since flow is assumed uniform over surface 1, then

$$\frac{\partial}{\partial t}(\rho\forall) + \rho_1 V_1 A_1 = 0 \qquad \text{or} \qquad \frac{\partial}{\partial t}(\rho\forall) = -\rho_1 V_1 A_1$$

Since the volume, $\forall$, of the tank is not a function of time,

$$\forall\frac{\partial\rho}{\partial t} = -\rho_1 V_1 A_1$$

and

$$\frac{\partial\rho}{\partial t} = -\frac{\rho_1 V_1 A_1}{\forall}$$

At $t = 0$,

$$\frac{\partial\rho}{\partial t} = -6\,\frac{\mathrm{kg}}{\mathrm{m}^3} \times 300\,\frac{\mathrm{m}}{\mathrm{s}} \times 65\,\mathrm{mm}^2 \times \frac{1}{0.05\,\mathrm{m}^3} \times \frac{\mathrm{m}^2}{10^6\,\mathrm{mm}^2}$$

$$\frac{\partial\rho}{\partial t} = -2.34\,(\mathrm{kg/m^3})/\mathrm{s} \xleftarrow{\{\text{The density is decreasing.}\}} \frac{\partial\rho}{\partial t}$$

This problem demonstrates use of the conservation of mass equation for unsteady flow problems.

4.4 Momentum Equation for Inertial Control Volume

We now wish to obtain a control volume form of Newton's second law. We use the same procedure we just used for mass conservation, with one note of caution: the control volume coordinates (with respect to which we measure all velocities) are inertial; that is, the control volume coordinates xyz are either at rest or moving at constant speed with respect to an "absolute" set of coordinates XYZ. (Sections 4.6 and 4.7 will analyze noninertial control volumes.) We begin with the mathematical formulation for a system and then use Eq. 4.10 to go from the system to the control volume formulation.

Recall that Newton's second law for a system moving relative to an inertial coordinate system was given by Eq. 4.2a as

$$\vec{F} = \left.\frac{d\vec{P}}{dt}\right)_{\text{system}} \tag{4.2a}$$

where the linear momentum of the system is given by

$$\vec{P}_{\text{system}} = \int_{M(\text{system})} \vec{V}\,dm = \int_{\mathcal{V}(\text{system})} \vec{V}\,\rho\,d\mathcal{V} \tag{4.2b}$$

and the resultant force, $\vec{F}$, includes all surface and body forces acting on the system,

$$\vec{F} = \vec{F}_S + \vec{F}_B$$

The system and control volume formulations are related using Eq. 4.10,

$$\left.\frac{dN}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} \eta\,\rho\,d\mathcal{V} + \int_{\text{CS}} \eta\,\rho\vec{V}\cdot d\vec{A} \tag{4.10}$$

To derive the control volume formulation of Newton's second law, we set

$$N = \vec{P} \qquad \text{and} \qquad \eta = \vec{V}$$

From Eq. 4.10, with this substitution, we obtain

$$\left.\frac{d\vec{P}}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} \vec{V}\,\rho\,d\mathcal{V} + \int_{\text{CS}} \vec{V}\rho\vec{V}\cdot d\vec{A} \tag{4.16}$$

From Eq. 4.2a

$$\left.\frac{d\vec{P}}{dt}\right)_{\text{system}} = \vec{F})_{\text{on system}} \tag{4.2a}$$

Since, in deriving Eq. 4.10, the system and the control volume coincided at t_0, then

$$\vec{F})_{\text{on system}} = \vec{F})_{\text{on control volume}}$$

In light of this, Eqs. 4.2a and 4.16 may be combined to yield the control volume formulation of Newton's second law for a nonaccelerating control volume

$$\vec{F} = \vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t}\int_{\text{CV}} \vec{V}\,\rho d\mathcal{V} + \int_{\text{CS}} \vec{V}\,\rho\vec{V}\cdot d\vec{A} \tag{4.17a}$$

For cases when we have uniform flow at each inlet and exit, we can use

$$\vec{F} = \vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t}\int_{\text{CV}} \vec{V}\,\rho d\mathcal{V} + \sum_{\text{CS}} \vec{V}\,\rho\vec{V}\cdot\vec{A} \tag{4.17b}$$

Equations 4.17a and 4.17b are our (nonaccelerating) control volume forms of Newton's second law. It states that the total force (due to surface and body forces) acting on the control volume leads to a rate of change of momentum within the control volume (the volume integral) and/or a net rate at which momentum is leaving the control volume through the control surface.

We must be a little careful in applying Eqs. 4.17. The first step will always be to carefully choose a control volume and its control surface so that we can evaluate the volume integral and the surface integral (or summation); each inlet and exit should be

carefully labeled, as should the external forces acting. In fluid mechanics the body force is usually gravity, so

$$\vec{F}_B = \int_{\text{CV}} \rho\vec{g}\, d\mathcal{V} = \vec{W}_{\text{CV}} = M\vec{g}$$

where $\vec{g}$ is the acceleration of gravity and $\vec{W}_{\text{CV}}$ is the instantaneous weight of the entire control volume. In many applications the surface force is due to pressure,

$$\vec{F}_S = \int_A -p d\vec{A}$$

Note that the minus sign is to ensure that we always compute pressure forces acting *onto* the control surface (recall $d\vec{A}$ was chosen to be a vector pointing *out* of the control volume). It is worth stressing that *even at points on the surface that have an outflow*, the pressure force acts *onto* the control volume.

In Eqs. 4.17 we must also be careful in evaluating $\int_{\text{CS}}\vec{V}\rho\vec{V}\cdot d\vec{A}$ or $\Sigma_{\text{CS}}\vec{V}\rho\vec{V}\cdot\vec{A}$ (this may be easier to do if we write them with the implied parentheses, $\int_{\text{CS}}\vec{V}\rho(\vec{V}\cdot d\vec{A})$ or $\Sigma_{\text{CS}}\vec{V}\rho(\vec{V}\cdot\vec{A})$). The velocity $\vec{V}$ must be measured with respect to the control volume coordinates xyz, with the appropriate signs for its vector components u, v, and w; recall also that the scalar product will be positive for outflow and negative for inflow (refer to Fig. 4.3).

The momentum equation (Eqs. 4.17) is a vector equation. We will usually write the three scalar components, as measured in the xyz coordinates of the control volume,

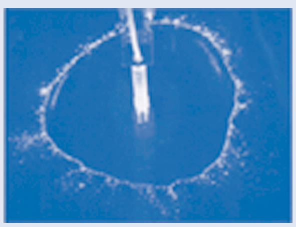
$$F_x = F_{S_x} + F_{B_x} = \frac{\partial}{\partial t}\int_{\text{CV}} u\,\rho\, d\mathcal{V} + \int_{\text{CS}} u\,\rho\vec{V}\cdot d\vec{A} \tag{4.18a}$$

$$F_y = F_{S_y} + F_{B_y} = \frac{\partial}{\partial t}\int_{\text{CV}} v\,\rho\, d\mathcal{V} + \int_{\text{CS}} v\,\rho\vec{V}\cdot d\vec{A} \tag{4.18b}$$

$$F_z = F_{S_z} + F_{B_z} = \frac{\partial}{\partial t}\int_{\text{CV}} w\,\rho\, d\mathcal{V} + \int_{\text{CS}} w\,\rho\vec{V}\cdot d\vec{A} \tag{4.18c}$$

VIDEO

Momentum Effect: A Jet Impacting a Surface.



or, for uniform flow at each inlet and exit,

$$F_x = F_{S_x} + F_{B_x} = \frac{\partial}{\partial t}\int_{\text{CV}} u\,\rho\, d\mathcal{V} + \sum_{\text{CS}} u\,\rho\vec{V}\cdot\vec{A} \tag{4.18d}$$

$$F_y = F_{S_y} + F_{B_y} = \frac{\partial}{\partial t}\int_{\text{CV}} v\,\rho\, d\mathcal{V} + \sum_{\text{CS}} v\,\rho\vec{V}\cdot\vec{A} \tag{4.18e}$$

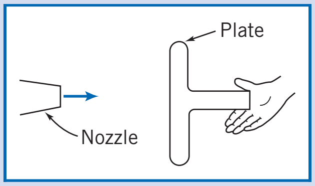
$$F_z = F_{S_z} + F_{B_z} = \frac{\partial}{\partial t}\int_{\text{CV}} w\,\rho\, d\mathcal{V} + \sum_{\text{CS}} w\,\rho\vec{V}\cdot\vec{A} \tag{4.18f}$$

Note that, as we found for the mass conservation equation (Eq. 4.12), for steady flow the first term on the right in Eqs. 4.17 and 4.18 is zero.

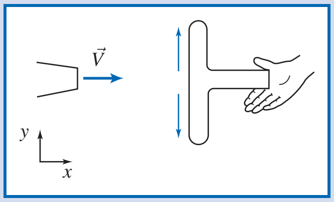
We will now look at five Examples to illustrate some features of the various forms of the momentum equation for a control volume. Example 4.4 demonstrates how intelligent choice of the control volume can simplify analysis of a problem, Example 4.5 involves a problem in which we have significant body forces, Example 4.6 explains how to simplify surface force evaluations by working in gage pressures, Example 4.7 involves nonuniform surface forces, and Example 4.8 involves a problem in which we have unsteady flow.

Example 4.4 CHOICE OF CONTROL VOLUME FOR MOMENTUM ANALYSIS

Water from a stationary nozzle strikes a flat plate as shown. The water leaves the nozzle at 15 m/s; the nozzle area is 0.01 m^2. Assuming the water is directed normal to the plate, and flows along the plate, determine the horizontal force you need to resist to hold it in place.



Given: Water from a stationary nozzle is directed normal to the plate; subsequent flow is parallel to plate.

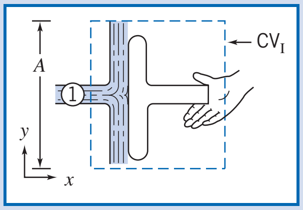


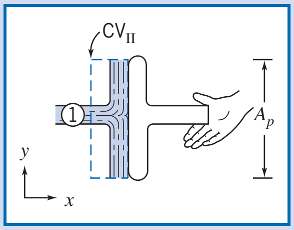
$$\text{Jet velocity}, \vec{V} = 15\hat{i}\text{ m/s}$$

$$\text{Nozzle area}, A_n = 0.01\text{ m}^2$$

Find: Horizontal force on your hand.

Solution: We chose a coordinate system in defining the problem above. We must now choose a suitable control volume. Two possible choices are shown by the dashed lines below.





In both cases, water from the nozzle crosses the control surface through area A_1 (assumed equal to the nozzle area) and is assumed to leave the control volume tangent to the plate surface in the $+y$ or $-y$ direction. Before trying to decide which is the "best" control volume to use, let us write the governing equations.

$$\vec{F} = \vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t}\int_{\text{CV}} \vec{V}\rho d\forall + \int_{\text{CS}} \vec{V}\rho\vec{V}\cdot d\vec{A} \qquad \text{and} \qquad \frac{\partial}{\partial t}\int_{\text{CV}} \rho\, d\forall + \int_{\text{CS}} \rho\vec{V}\cdot d\vec{A} = 0$$

Assumptions:
(1) Steady flow.
(2) Incompressible flow.
(3) Uniform flow at each section where fluid crosses the CV boundaries.

Regardless of our choice of control volume, assumptions (1), (2), and (3) lead to

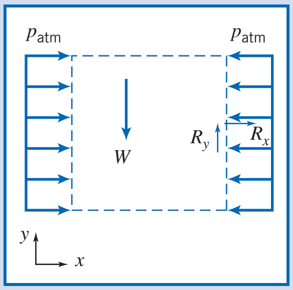
$$\vec{F} = \vec{F}_S + \vec{F}_B = \sum_{\text{CS}} \vec{V}\rho\vec{V}\cdot\vec{A} \qquad \text{and} \qquad \sum_{\text{CS}} \rho\vec{V}\cdot\vec{A} = 0$$

Evaluating the momentum flux term will lead to the same result for both control volumes. We should choose the control volume that allows the most straightforward evaluation of the forces.

Remember in applying the momentum equation that the force, $\vec{F}$, represents all forces acting *on* the control volume.

Let us solve the problem using each of the control volumes.

CV_I



The control volume has been selected so that the area of the left surface is equal to the area of the right surface. Denote this area by A.

The control volume cuts through your hand. We denote the components of the reaction force of your hand on the control volume as R_x and R_y and assume both to be positive. (The force of the control volume on your hand is equal and opposite to R_x and R_y.)

Atmospheric pressure acts on all surfaces of the control volume. Note that *the pressure in a free jet is ambient*, i.e., in this case atmospheric. (The distributed force due to atmospheric pressure has been shown on the vertical faces only.)

The body force on the control volume is denoted as W.

Since we are looking for the horizontal force, we write the x component of the steady flow momentum equation

$$F_{S_x} + F_{B_x} = \sum_{\mathrm{CS}} u\,\rho\vec{V}\cdot\vec{A}$$

There are no body forces in the x direction, so $F_{B_x} = 0$, and

$$F_{S_x} = \sum_{\mathrm{CS}} u\,\rho\vec{V}\cdot\vec{A}$$

To evaluate F_{S_x}, we must include all surface forces acting on the control volume

$$F_{S_x} = \underbrace{p_{\mathrm{atm}}A}_{\substack{\text{force due to atmospheric}\\ \text{pressure acts to right}\\ \text{(positive direction)}\\ \text{on left surface}}} - \underbrace{p_{\mathrm{atm}}A}_{\substack{\text{force due to atmospheric}\\ \text{pressure acts to left}\\ \text{(negative direction)}\\ \text{on right surface}}} + \underbrace{R_x}_{\substack{\text{force of your hand on}\\ \text{control volume}\\ \text{(assumed positive)}}}$$

Consequently, $F_{S_x} = R_x$, and

$$R_x = \sum_{\mathrm{CS}} u\,\rho\vec{V}\cdot\vec{A} = u\,\rho\vec{V}\cdot\vec{A}\big|_1 \qquad \{\text{For top and bottom surfaces, } u = 0\}$$

$$R_x = -u_1\,\rho V_1 A_1 \qquad \{\text{At } (1),\ \rho\vec{V}_1\cdot\vec{A}_1 = \rho(-V_1A_1) \text{ since } \vec{V}_1 \text{ and } \vec{A}_1 \text{ are } 180^\circ \text{ apart. Note that } u_1 = V_1\}$$

$$R_x = -15\,\frac{\mathrm{m}}{\mathrm{s}} \times 999\,\frac{\mathrm{kg}}{\mathrm{m}^3} \times 15\,\frac{\mathrm{m}}{\mathrm{s}} \times 0.01\ \mathrm{m}^2 \times \frac{\mathrm{N}\cdot\mathrm{s}^2}{\mathrm{kg}\cdot\mathrm{m}} \qquad \{u_1 = 15\ \mathrm{m/s}\}$$

$$R_x = -2.25\ \mathrm{kN} \qquad \{R_x \text{ acts opposite to positive direction assumed.}\}$$

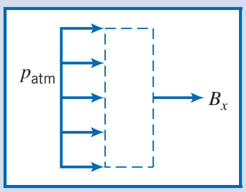
The horizontal force on your hand is

$$K_x = -R_x = 2.25\ \mathrm{kN} \longleftarrow \{\text{force on your hand acts to the right}\} \qquad K_x$$

CV$_{\mathrm{II}}$ with Horizontal Forces Shown

The control volume has been selected so the areas of the left surface and of the right surface are equal to the area of the plate. Denote this area by A_p.

The control volume is in contact with the plate over the entire plate surface. We denote the horizontal reaction force from the plate on the control volume as B_x (and assume it to be positive).



Atmospheric pressure acts on the left surface of the control volume (and on the two horizontal surfaces).

The body force on this control volume has no component in the x direction.

Then the x component of the momentum equation,

$$F_{S_x} = \sum_{\mathrm{CS}} u\,\rho\vec{V}\cdot\vec{A}$$

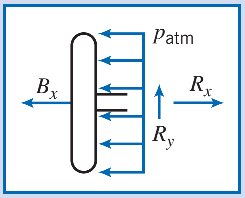
yields

$$F_{S_x} = p_{\mathrm{atm}}\,A_p + B_x = u\,\rho\vec{V}\cdot\vec{A}\big|_1 = -u_1 V_1 A_1 = -2.25\ \mathrm{kN}$$

Then

$$B_x = -p_{\text{atm}} A_p - 2.25\,\text{kN}$$

To determine the net force on the plate, we need a free-body diagram of the plate:



$$\begin{aligned}\sum F_x &= 0 = -B_x - p_{\text{atm}}A_p + R_x\\ R_x &= p_{\text{atm}}A_p + B_x\\ R_x &= p_{\text{atm}}A_p + (-p_{\text{atm}}A_p - 2.25\,\text{kN}) = -2.25\,\text{kN}\end{aligned}$$

Then the horizontal force on your hand is $K_x = -R_x = 2.25$ kN.

Note that the choice of CV_{II} meant we needed an additional free-body diagram. In general it is best to select the control volume so that the force sought acts explicitly on the control volume.

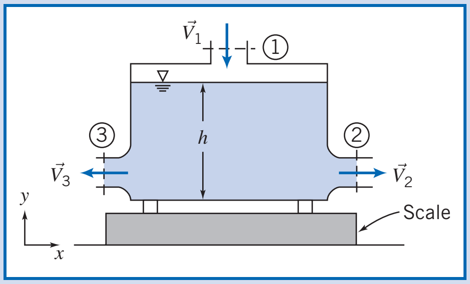
Notes:

- ✓ This problem demonstrates how thoughtful choice of the control volume can simplify use of the momentum equation.
- ✓ The analysis would have been greatly simplified if we had worked in gage pressures (see Example 4.6).
- ✓ For this problem the force generated was entirely due to the plate absorbing the jet's horizontal momentum.

Example 4.5 TANK ON SCALE: BODY FORCE

A metal container 2 ft high, with an inside cross-sectional area of 1 ft^2, weighs 5 lbf when empty. The container is placed on a scale and water flows in through an opening in the top and out through the two equal-area openings in the sides, as shown in the diagram. Under steady flow conditions, the height of the water in the tank is $h = 1.9$ ft.

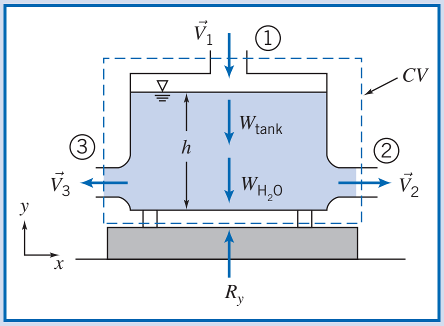
$$\begin{aligned}A_1 &= 0.1\text{ft}^2\\ \vec{V}_1 &= -10\hat{j}\,\text{ft/s}\\ A_2 &= A_3 = 0.1\,\text{ft}^2\end{aligned}$$



Your boss claims that the scale will read the weight of the volume of water in the tank plus the tank weight, i.e., that we can treat this as a simple statics problem. You disagree, claiming that a fluid flow analysis is required. Who is right, and what does the scale indicate?

Given: Metal container, of height 2 ft and cross-sectional area $A = 1$ ft^2, weighs 5 lbf when empty. Container rests on scale. Under steady flow conditions water depth is $h = 1.9$ ft. Water enters vertically at section 1 and leaves horizontally through sections 2 and 3

$$\begin{aligned}A_1 &= 0.1\,\text{ft}^2\\ \vec{V}_1 &= -10\hat{j}\,\text{ft/s}\\ A_2 &= A_3 = 0.1\,\text{ft}^2\end{aligned}$$



Find: Scale reading.

Solution:

Choose a control volume as shown; R_y is the force of the scale on the control volume (exerted on the control volume through the supports) and is assumed positive.

The weight of the tank is designated W_{tank}; the weight of the water in the tank is W_{H_2O}.

Atmospheric pressure acts uniformly on the entire control surface, and therefore has no net effect on the control volume. Because of this null effect we have not shown the pressure distribution in the diagram.

Governing equations: The general control volume momentum and mass conservation equations are Eqs. 4.17 and 4.12, respectively,

$$\vec{F}_S + \vec{F}_B = \underbrace{\cancel{\frac{\partial}{\partial t}}\int_{\text{CV}} \vec{V}\rho\, d\forall}_{=0(1)} + \int_{\text{CS}} \vec{V}\rho\vec{V}\cdot d\vec{A}$$

$$\underbrace{\cancel{\frac{\partial}{\partial t}}\int_{\text{CV}} \rho\, d\forall}_{=0(1)} + \int_{\text{CS}} \rho\vec{V}\cdot d\vec{A} = 0$$

Note that we usually start with the simplest forms (based on the problem assumptions, e.g., steady flow) of the mass conservation and momentum equations. However, in this problem, for illustration purposes, we start with the most general forms of the equations.

Assumptions:
(1) Steady flow (given).
(2) Incompressible flow.
(3) Uniform flow at each section where fluid crosses the CV boundaries.

We are only interested in the y component of the momentum equation

$$F_{S_y} + F_{B_y} = \int_{\text{CS}} v\,\rho\vec{V}\cdot d\vec{A} \tag{1}$$

$$F_{S_y} = R_y \qquad \{\text{There is no net force due to atmosphere pressure.}\}$$

$$F_{B_y} = -W_{\text{tank}} - W_{H_2O} \qquad \{\text{Both body forces act in negative } y \text{ direction.}\}$$

$$W_{H_2O} = \rho g\forall = \gamma A h$$

$$\int_{\text{CS}} v\,\rho\vec{V}\cdot d\vec{A} = \int_{A_1} v\,\rho\vec{V}\cdot d\vec{A} = \int_{A_1} v(-\rho V_1 dA_1) \qquad \left\{\begin{array}{l}\vec{V}\cdot d\vec{A} \text{ is negative at } 1\\ v = 0 \text{ at sections } 2 \text{ and } 3\end{array}\right\}$$

$$= v_1(-\rho V_1 A_1) \qquad \left\{\begin{array}{l}\text{We are assuming uniform}\\ \text{properties at } 1\end{array}\right\}$$

Using these results in Eq. 1 gives

$$R_y - W_{\text{tank}} - \gamma A h = v_1(-\rho V_1 A_1)$$

Note that v_1 is the y component of the velocity, so that $v_1 = -V_1$, where we recall that $V_1 = 10$ ft/s is the magnitude of velocity $\vec{V}_1$. Hence, solving for R_y,

$$R_y = W_{\text{tank}} + \gamma A h + \rho V_1^2 A_1$$

$$= 5\text{ lbf} + 62.4\frac{\text{lbf}}{\text{ft}^3} \times 1\text{ ft}^2 \times 1.9\text{ ft} + 1.94\frac{\text{slug}}{\text{ft}^3} \times 100\frac{\text{ft}^2}{\text{s}^2} \times 0.1\text{ ft}^2 \times \frac{\text{lbf}\cdot\text{s}^2}{\text{slug}\cdot\text{ft}}$$

$$= 5\text{ lbf} + 118.6\text{ lbf} + 19.4\text{ lbf}$$

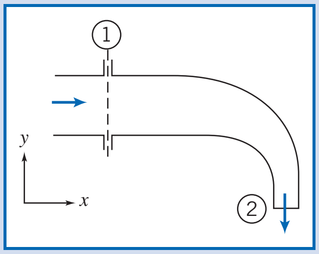
$$R_y = 143\text{ lbf} \longleftarrow R_y$$

Note that this is the force of the scale on the control volume; it is also the reading on the scale. We can see that the scale reading is due to: the tank weight (5 lbf), the weight of water instantaneously in the tank (118.6 lbf), and the force involved in absorbing the downward momentum of the fluid at section 1 (19.4 lbf). Hence your boss is wrong—neglecting the momentum results in an error of almost 15%.

This problem illustrates use of the momentum equation including significant body forces.

Example 4.6 FLOW THROUGH ELBOW: USE OF GAGE PRESSURES

Water flows steadily through the 90° reducing elbow shown in the diagram. At the inlet to the elbow, the absolute pressure is 220 kPa and the cross-sectional area is 0.01 m^2. At the outlet, the cross-sectional area is 0.0025 m^2 and the velocity is 16 m/s. The elbow discharges to the atmosphere. Determine the force required to hold the elbow in place.



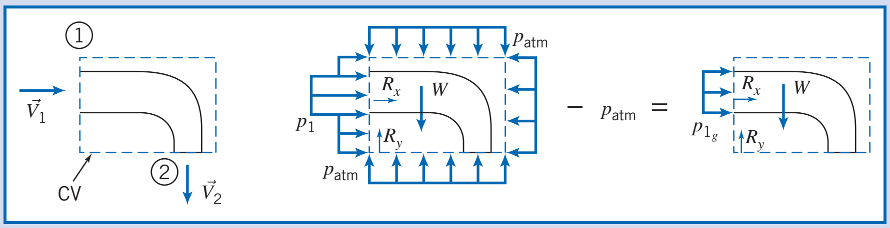
Given: Steady flow of water through 90° reducing elbow.

$$p_1 = 220\,\text{kPa (abs)} \qquad A_1 = 0.01\,\text{m}^2 \qquad \vec{V}_2 = -16\,\hat{j}\,\text{m/s} \qquad A_2 = 0.0025\,\text{m}^2$$

Find: Force required to hold elbow in place.

Solution:
Choose a fixed control volume as shown. Note that we have several surface force computations: p_1 on area A_1 and p_{atm} everywhere else. The exit at section 2 is to a free jet, and so at ambient (i.e., atmospheric) pressure. We can use a simplification here: If we subtract p_{atm} from the entire surface (a null effect as far as forces are concerned) we can work in gage pressures, as shown.

Note that since the elbow is anchored to the supply line, in addition to the reaction forces R_x and R_y (shown), there would also be a reaction moment (not shown).



Governing equations:

$$\vec{F} = \vec{F}_S + \vec{F}_B = \overset{=0(4)}{\cancel{\frac{\partial}{\partial t}\int_{\text{CV}} \vec{V}\rho\, d\forall}} + \int_{\text{CS}} \vec{V}\rho\vec{V}\cdot d\vec{A}$$

$$\overset{=0(4)}{\cancel{\frac{\partial}{\partial t}\int_{\text{CV}} \rho\, d\forall}} + \int_{\text{CS}} \rho\vec{V}\cdot d\vec{A} = 0$$

Assumptions:
(1) Uniform flow at each section.
(2) Atmospheric pressure, $p_{\text{atm}} = 101$ kPa (abs).
(3) Incompressible flow.
(4) Steady flow (given).
(5) Neglect weight of elbow and water in elbow.

Once again (although we didn't need to) we started with the most general form of the governing equations. Writing the x component of the momentum equation results in

$$F_{S_x} = \int_{\text{CS}} u\rho \vec{V} \cdot d\vec{A} = \int_{A_1} u\rho \vec{V} \cdot d\vec{A} \qquad \{F_{B_x} = 0 \quad \text{and} \quad u_2 = 0\}$$

$$p_{1_g} A_1 + R_x = \int_{A_1} u\, \rho \vec{V} \cdot d\vec{A}$$

so

$$R_x = -p_{1_g} A_1 + \int_{A_1} u\, \rho \vec{V} \cdot d\vec{A}$$

$$= -p_{1_g} A_1 + u_1(-\rho V_1 A_1)$$

$$R_x = -p_{1_g} A_1 - \rho V_1^2 A_1$$

Note that u_1 is the x component of the velocity, so that $u_1 = V_1$. To find V_1, use the mass conservation equation:

$$\int_{\text{CS}} \rho \vec{V} \cdot d\vec{A} = \int_{A_1} \rho \vec{V} \cdot d\vec{A} + \int_{A_2} \rho \vec{V} \cdot d\vec{A} = 0$$

$$\therefore (-\rho V_1 A_1) + (\rho V_2 A_2) = 0$$

and

$$V_1 = V_2 \frac{A_2}{A_1} = 16 \frac{\text{m}}{\text{s}} \times \frac{0.0025}{0.01} = 4 \text{ m/s}$$

We can now compute R_x

$$R_x = -p_{1_g} A_1 - \rho V_1^2 A_1$$

$$= -1.19 \times 10^5 \frac{\text{N}}{\text{m}^2} \times 0.01 \text{ m}^2 - 999 \frac{\text{kg}}{\text{m}^3} \times 16 \frac{\text{m}^2}{\text{s}^2} \times 0.01 \text{ m}^2 \times \frac{\text{N} \cdot \text{s}^2}{\text{kg} \cdot \text{m}}$$

$$R_x = -1.35 \text{ kN} \xleftarrow{\hspace{6em}} R_x$$

Writing the y component of the momentum equation gives

$$F_{S_y} + F_{B_y} = R_y + F_{B_y} = \int_{\text{CS}} v\, \rho \vec{V} \cdot d\vec{A} = \int_{A_2} v\, \rho \vec{V} \cdot d\vec{A} \qquad \{v_1 = 0\}$$

or

$$R_y = -F_{B_y} + \int_{A_2} v\, \rho \vec{V} \cdot d\vec{A}$$

$$= -F_{B_y} + v_2(\rho V_2 A_2)$$

$$R_y = -F_{B_y} - \rho V_2^2 A_2$$

Note that v_2 is the y component of the velocity, so that $v_2 = -V_2$, where V_2 is the magnitude of the exit velocity.

Substituting known values

$$R_y = -F_{B_y} + -\rho V_2^2 A_2$$

$$= -F_{B_y} - 999 \frac{\text{kg}}{\text{m}^3} \times (16)^2 \frac{\text{m}^2}{\text{s}^2} \times 0.0025 \text{ m}^2 \times \frac{\text{N} \cdot \text{s}^2}{\text{kg} \cdot \text{m}}$$

$$= -F_{B_y} - 639 \text{ N} \xleftarrow{\hspace{6em}} R_y$$

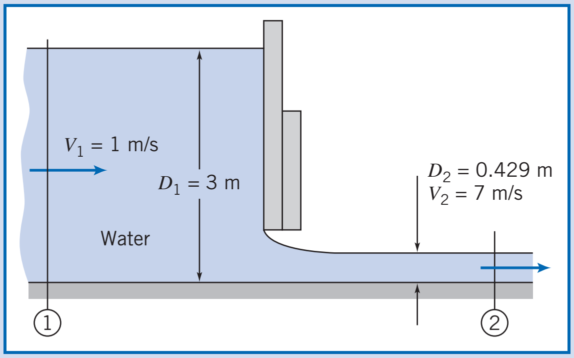
Neglecting F_{B_y} gives

$$R_y = -639 \text{ N} \xleftarrow{\hspace{6em}} R_y$$

This problem illustrates how using gage pressures simplifies evaluation of the surface forces in the momentum equation.

Example 4.7 FLOW UNDER A SLUICE GATE: HYDROSTATIC PRESSURE FORCE

Water in an open channel is held in by a sluice gate. Compare the horizontal force of the water on the gate (a) when the gate is closed and (b) when it is open (assuming steady flow, as shown). Assume the flow at sections 1 and 2 is incompressible and uniform, and that (because the streamlines are straight there) the pressure distributions are hydrostatic.

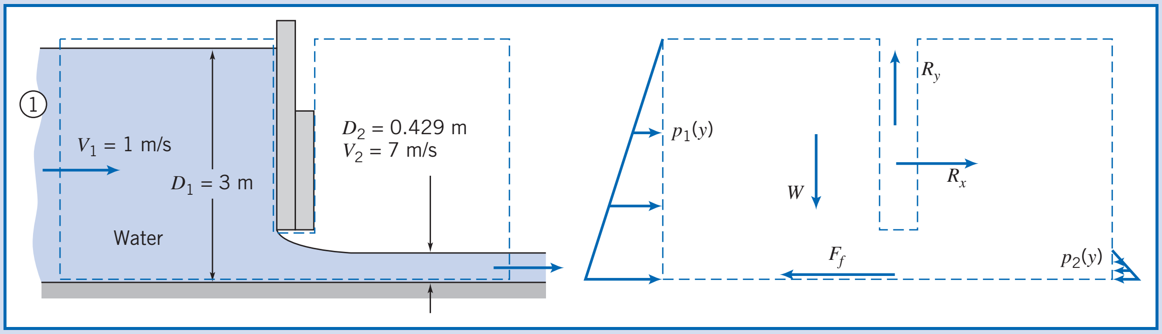


Given: Flow under sluice gate. Width $= w$.

Find: Horizontal force (per unit width) on the closed and open gate.

Solution:

Choose a control volume as shown for the open gate. Note that it is much simpler to work in gage pressures, as we learned in Example 4.6.



The forces acting on the control volume include:

- Force of gravity W.
- Friction force F_f.
- Components R_x and R_y of reaction force from gate.
- Hydrostatic pressure distribution on vertical surfaces, assumption (6).
- Pressure distribution $p_b(x)$ along bottom surface (not shown).

Apply the x component of the momentum equation.

Governing equation:

$$F_{S_x} + \overset{=0(2)}{\cancel{F_{B_x}}} = \overset{=0(3)}{\cancel{\frac{\partial}{\partial t}\int_{\mathrm{CV}} u\,\rho\, d\forall}} + \int_{\mathrm{CS}} u\,\rho \vec{V}\cdot d\vec{A}$$

Assumptions:
(1) F_f negligible (neglect friction on channel bottom).
(2) $F_{B_x} = 0$.
(3) Steady flow.
(4) Incompressible flow (given).
(5) Uniform flow at each section (given).
(6) Hydrostatic pressure distributions at 1 and 2 (given).

Then

$$F_{S_x} = F_{R_1} + F_{R_2} + R_x = u_1(-\rho V_1 w D_1) + u_2(\rho V_2 w D_2)$$

The surface forces acting on the CV are due to the pressure distributions and the unknown force R_x. From assumption (6), we can integrate the gage pressure distributions on each side to compute the hydrostatic forces F_{R_1} and F_{R_2},

$$F_{R_1} = \int_0^{D_1} p_1\, dA = w\int_0^{D_1} \rho g y\, dy = \rho g w \frac{y^2}{2}\bigg|_0^{D_1} = \frac{1}{2}\rho g w D_1^2$$

where y is measured downward from the free surface of location ①, and

$$F_{R_2} = \int_0^{D_2} p_2\, dA = w\int_0^{D_2} \rho g y\, dy = \rho g w \frac{y^2}{2}\bigg|_0^{D_2} = \frac{1}{2}\rho g w D_2^2$$

where y is measured downward from the free surface of location ②. (Note that we could have used the hydrostatic force equation, Eq. 3.10b, directly to obtain these forces.)

Evaluating F_{S_x} gives

$$F_{S_x} = R_x + \frac{\rho g w}{2}(D_1^2 - D_2^2)$$

Substituting into the momentum equation, with $u_1 = V_1$ and $u_2 = V_2$, gives

$$R_x + \frac{\rho g w}{2}(D_1^2 - D_2^2) = -\rho V_1^2 w D_1 + \rho V_2^2 w D_2$$

or

$$R_x = \rho w (V_2^2 D_2 - V_1^2 D_1) - \frac{\rho g w}{2}(D_1^2 - D_2^2)$$

The second term on the right is the net hydrostatic force on the gate; the first term "corrects" this (and leads to a smaller net force) for the case when the gate is open. What is the nature of this "correction"? The pressure in the fluid far away from the gate in either direction is indeed hydrostatic, but consider the flow close to the gate: Because we have significant velocity variations here (in magnitude and direction), the pressure distributions deviate significantly from hydrostatic—for example, as the fluid accelerates under the gate there will be a significant pressure drop on the lower left side of the gate. Deriving this pressure field would be a difficult task, but by careful choice of our CV we have avoided having to do so!

We can now compute the horizontal force per unit width,

$$\begin{aligned}
\frac{R_x}{w} &= \rho(V_2^2 D_2 - V_1^2 D_1) - \frac{\rho g}{2}(D_1^2 - D_2^2)\\
&= 999\,\frac{\text{kg}}{\text{m}^3} \times \left[(7)^2(0.429) - (1)^2(3)\right]\frac{\text{m}^2}{\text{s}^2}\,\text{m} \times \frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}}\\
&\quad - \frac{1}{2} \times 999\,\frac{\text{kg}}{\text{m}^3} \times 9.81\,\frac{\text{m}}{\text{s}^2} \times [(3)^2 - (0.429)^2]\text{m}^2 \times \frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}}\\
\frac{R_x}{w} &= 18.0\ \text{kN/m} - 43.2\ \text{kN/m}\\
\frac{R_x}{w} &= -25.2\ \text{kN/m}
\end{aligned}$$

R_x is the external force acting on the control volume, applied to the CV by the gate. Therefore, the force of the water on the gate is K_x, where $K_x = -R_x$. Thus,

$$\frac{K_x}{w} = -\frac{R_x}{w} = 25.2\ \text{kN/m} \longleftarrow \frac{K_x}{w}$$

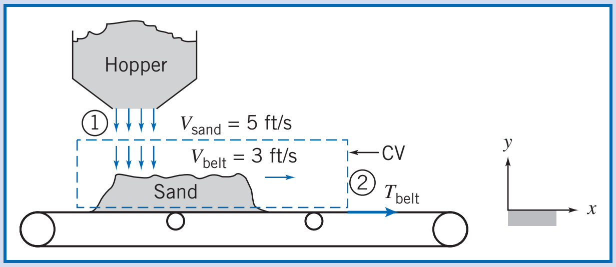
This force can be compared to the force on the closed gate of 44.1 kN (obtained from the second term on the right in the equation above, evaluated with D_2 set to zero because for the closed gate there is no fluid on the right of the gate)—the force on the open gate is significantly less as the water accelerates out under the gate.

This problem illustrates the application of the momentum equation to a control volume for which the pressure is not uniform on the control surface.

Example 4.8 CONVEYOR BELT FILLING: RATE OF CHANGE OF MOMENTUM IN CONTROL VOLUME

A horizontal conveyor belt moving at 3 ft/s receives sand from a hopper. The sand falls vertically from the hopper to the belt at a speed of 5 ft/s and a flow rate of 500 lbm/s (the density of sand is approximately 2700 lbm/cubic yard). The conveyor belt is initially empty but begins to fill with sand. If friction in the drive system and rollers is negligible, find the tension required to pull the belt while the conveyor is filling.

Given: Conveyor and hopper shown in sketch.



Find: T_{belt} at the instant shown.

Solution: Use the control volume and coordinates shown. Apply the x component of the momentum equation.

Governing equations:

$$F_{S_x} + \cancelto{0(2)}{F_{B_x}} = \frac{\partial}{\partial t}\int_{\text{CV}} u\,\rho\,d\forall + \int_{\text{CS}} u\,\rho\vec{V}\cdot d\vec{A} \qquad \frac{\partial}{\partial t}\int_{\text{CV}} \rho\,d\forall + \int_{\text{CS}} \rho\vec{V}\cdot d\vec{A} = 0$$

Assumptions:
(1) $F_{S_x} = T_{\text{belt}} = T$.
(2) $F_{B_x} = 0$.
(3) Uniform flow at section 1.
(4) All sand on belt moves with $V_{\text{belt}} = V_b$.

Then

$$T = \frac{\partial}{\partial t}\int_{\text{CV}} u\rho\,d\forall + u_1(-\rho V_1 A_1) + u_2(\rho V_2 A_2)$$

Since $u_1 = 0$, and there is no flow at section 2,

$$T = \frac{\partial}{\partial t}\int_{\text{CV}} u\rho\,d\forall$$

From assumption (4), inside the CV, $u = V_b =$ constant, and hence

$$T = V_b\frac{\partial}{\partial t}\int_{\text{CV}} \rho\,d\forall = V_b\frac{\partial M_s}{\partial t}$$

where M_s is the mass of sand on the belt (inside the control volume). This result is perhaps not surprising—the tension in the belt is the force required to increase the momentum inside the CV (which is increasing because even though the velocity of the mass in the CV is constant, the mass is not). From the continuity equation,

$$\frac{\partial}{\partial t}\int_{\text{CV}} \rho\,d\forall = \frac{\partial}{\partial t}M_s = -\int_{\text{CS}} \rho\vec{V}\cdot d\vec{A} = \dot{m}_s = 500\,\text{lbm/s}$$

Then

$$T = V_b\dot{m}_s = 3\,\frac{\text{ft}}{\text{s}} \times 500\,\frac{\text{lbm}}{s} \times \frac{\text{slug}}{32.2\,\text{lbm}} \times \frac{\text{lbf}\cdot\text{s}^2}{\text{slug}\cdot\text{ft}}$$

$$T = 46.6\,\text{lbf} \longleftarrow T$$

This problem illustrates application of the momentum equation to a control volume in which the momentum is changing.

*Differential Control Volume Analysis

The control volume approach, as we have seen in the previous examples, provides useful results when applied to a finite region.

If we apply the approach to a differential control volume, we can obtain differential equations describing a flow field. In this section, we will apply the conservation of mass and momentum equations to such a control volume to obtain a simple differential equation describing flow in a steady, incompressible, frictionless flow, and integrate it along a streamline to obtain the famous Bernoulli equation.

Let us apply the continuity and momentum equations to a steady incompressible flow without friction, as shown in Fig. 4.4. The control volume chosen is fixed in space and bounded by flow streamlines, and is thus an element of a stream tube. The length of the control volume is *ds*.

Because the control volume is bounded by streamlines, flow across the bounding surfaces occurs only at the end sections. These are located at coordinates s and $s + ds$, measured along the central streamline.

Properties at the inlet section are assigned arbitrary symbolic values. Properties at the outlet section are assumed to increase by differential amounts. Thus at $s + ds$, the flow speed is assumed to be $V_s + dV_s$, and so on. The differential changes, dp, dV_s, and dA, all are assumed to be positive in setting up the problem. (As in a free-body analysis in statics or dynamics, the actual algebraic sign of each differential change will be determined from the results of the analysis.)

Now let us apply the continuity equation and the s component of the momentum equation to the control volume of Fig. 4.4.

a. Continuity Equation

Basic equation:

$$\overbrace{\cancel{\frac{\partial}{\partial t}}\int_{\text{CV}} \rho \, d\mathcal{V}}^{=0(1)} + \int_{\text{CS}} \rho \vec{V} \cdot d\vec{A} = 0 \tag{4.12}$$

Assumptions: (1) Steady flow.
(2) No flow across bounding streamlines.
(3) Incompressible flow, $\rho = \text{constant}$.

Then

$$(-\rho V_s A) + \{\rho(V_s + dV_s)(A + dA)\} = 0$$

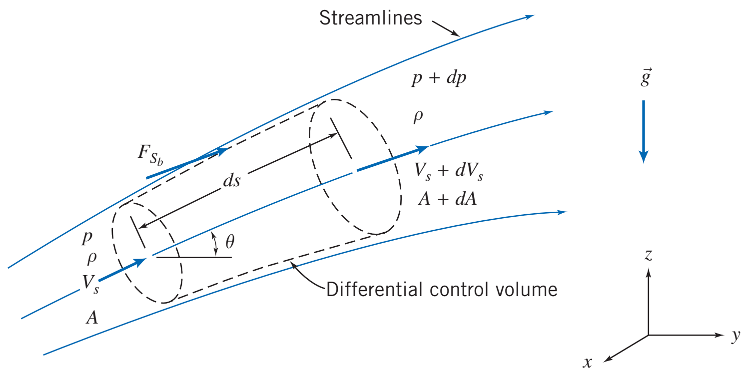


Fig. 4.4 Differential control volume for momentum analysis of flow through a stream tube.

*This section may be omitted without loss of continuity in the text material.

so

$$\rho(V_s + dV_s)(A + dA) = \rho V_s A \tag{4.19a}$$

On expanding the left side and simplifying, we obtain

$$V_s\,dA + A\,dV_s + dA\,dV_s = 0$$

But $dA\,dV_s$ is a product of differentials, which may be neglected compared with $V_s\,dA$ or $A\,dV_s$. Thus

$$V_s\,dA + A\,dV_s = 0 \tag{4.19b}$$

b. Streamwise Component of the Momentum Equation

Basic equation:
$$F_{S_s} + F_{B_s} = \overbrace{\cancel{\frac{\partial}{\partial t}}\int_{\text{CV}} u_s\,\rho\,d\Psi}^{=0(1)} + \int_{\text{CS}} u_s\,\rho\vec{V}\cdot d\vec{A} \tag{4.20}$$

Assumption: (4) No friction, so F_{S_b} is due to pressure forces only.

The surface force (due only to pressure) will have three terms:

$$F_{S_s} = pA - (p + dp)(A + dA) + \left(p + \frac{dp}{2}\right)dA \tag{4.21a}$$

The first and second terms in Eq. 4.21a are the pressure forces on the end faces of the control surface. The third term is F_{S_b}, the pressure force acting in the s direction on the bounding stream surface of the control volume. Its magnitude is the product of the average pressure acting on the stream surface, $p + \frac{1}{2}dp$, times the area component of the stream surface in the s direction, dA. Equation 4.21a simplifies to

$$F_{S_s} = -A\,dp - \frac{1}{2}dp\,dA \tag{4.21b}$$

The body force component in the s direction is

$$F_{B_s} = \rho g_s\,d\Psi = \rho(-g\sin\theta)\left(A + \frac{dA}{2}\right)ds$$

But $\sin\theta\,ds = dz$, so that

$$F_{B_s} = -\rho g\left(A + \frac{dA}{2}\right)dz \tag{4.21c}$$

The momentum flux will be

$$\int_{\text{CS}} u_s\,\rho\vec{V}\cdot d\vec{A} = V_s(-\rho V_s A) + (V_s + dV_s)\{\rho(V_s + dV_s)(A + dA)\}$$

since there is no mass flux across the bounding stream surfaces. The mass flux factors in parentheses and braces are equal from continuity, Eq. 4.19a, so

$$\int_{\text{CS}} u_s\,\rho\vec{V}\cdot d\vec{A} = V_s(-\rho V_s A) + (V_s + dV_s)(\rho V_s A) = \rho V_s A\,dV_s \tag{4.22}$$

Substituting Eqs. 4.21b, 4.21c, and 4.22 into Eq. 4.20 (the momentum equation) gives

$$-A\,dp - \frac{1}{2}dp\,dA - \rho gA\,dz - \frac{1}{2}\rho g\,dA\,dz = \rho V_s A\,dV_s$$

Dividing by ρA and noting that products of differentials are negligible compared with the remaining terms, we obtain

$$-\frac{dp}{\rho} - g\,dz = Vs\,dV_s = d\left(\frac{V_s^2}{2}\right)$$

or

$$\frac{dp}{\rho} + d\left(\frac{V_s^2}{2}\right) + g\,dz = 0 \tag{4.23}$$

Because the flow is incompressible, this equation may be integrated to obtain

$$\frac{P}{\rho} + \frac{V_s^2}{2} + gz = \text{constant} \tag{4.24}$$

or, dropping subscript s,

$$\frac{P}{\rho} + \frac{V_s^2}{2} + gz = \text{constant} \tag{4.24}$$

This equation is subject to the restrictions:

1. Steady flow.
2. No friction.
3. Flow along a streamline.
4. Incompressible flow.

We have derived one form of perhaps the most famous (and misused) equation in fluid mechanics—the Bernoulli equation. It can be used *only* when the four restrictions listed above apply, at least to reasonable accuracy! Although no real flow satisfies all these restrictions (especially the second), we can approximate the behavior of many flows with Eq. 4.24.

For example, the equation is widely used in aerodynamics to relate the pressure and velocity in a flow (e.g., it explains the lift of a subsonic wing). It could also be used to find the pressure at the inlet of the reducing elbow analyzed in Example 4.6 or to determine the velocity of water leaving the sluice gate of Example 4.7 (both of these flows approximately satisfy the four restrictions). On the other hand, Eq. 4.24 does *not* correctly describe the variation of water pressure in pipe flow. According to it, for a horizontal pipe of constant diameter, the pressure will be constant, but in fact the pressure drops significantly along the pipe—we will need most of Chapter 8 to explain this.

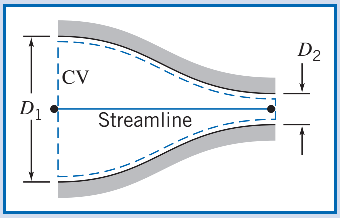
The Bernoulli equation, and the limits on its use, is so important we will derive it again and discuss its limitations in detail in Chapter 6.

Example 4.9 NOZZLE FLOW: APPLICATION OF BERNOULLI EQUATION

Water flows steadily through a horizontal nozzle, discharging to the atmosphere. At the nozzle inlet the diameter is D_1; at the nozzle outlet the diameter is D_2. Derive an expression for the minimum gage pressure required at the nozzle inlet to produce a given volume flow rate, Q. Evaluate the inlet gage pressure if $D_1 = 3.0$ in., $D_2 = 1.0$ in., and the desired flow rate is 0.7 ft^3/s.

Given: Steady flow of water through a horizontal nozzle, discharging to the atmosphere.

$$D_1 = 3.0\text{ in.} \qquad D_2 = 1.0\text{ in.} \qquad p_2 = p_{\text{atm}}$$



Find: (a) p_{1g} as a function of volume flow rate, Q.
(b) p_{1g} for $Q = 0.7\text{ ft}^3/\text{s}$.

Solution:

Governing equations:

$$\frac{p_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{p_2}{\rho} + \frac{V_2^2}{2} + gz_2$$

$$= 0(1)$$

$$\cancel{\frac{\partial}{\partial t}}\int_{\text{CV}} \rho\, d\mathcal{V} + \int_{\text{CS}} \rho \vec{V} \cdot d\vec{A} = 0$$

Assumptions:
(1) Steady flow (given).
(2) Incompressible flow.
(3) Frictionless flow.
(4) Flow along a streamline.
(5) $z_1 = z_2$.
(6) Uniform flow at sections 1 and 2.

Apply the Bernoulli equation along a streamline between points 1 and 2 to evaluate p_1. Then

$$p_{1_g} = p_1 - p_{\text{atm}} = p_1 - p_2 = \frac{\rho}{2}(V_2^2 - V_1^2) = \frac{\rho}{2}V_1^2\left[\left(\frac{V_2}{V_1}\right)^2 - 1\right]$$

Apply the continuity equation

$$(-\rho V_1 A_1) + (\rho V_2 A_2) = 0 \qquad \text{or} \qquad V_1 A_1 = V_2 A_2 = Q$$

so that

$$\frac{V_2}{V_1} = \frac{A_1}{A_2} \qquad \text{and} \qquad V_1 = \frac{Q}{A_1}$$

Then

$$p_{1_g} = \frac{\rho Q^2}{2A_1^2}\left[\left(\frac{A_1}{A_2}\right)^2 - 1\right]$$

Since $A = \pi D^2/4$, then

$$p_{1_g} = \frac{8\rho\, Q^2}{\pi^2\, D_1^4}\left[\left(\frac{D_1}{D_2}\right)^4 - 1\right] \longleftarrow p_{1_g}$$

(Note that for a given nozzle the pressure required is proportional to the square of the flow rate—not surpising since we have used Eq. 4.24, which shows that $p \sim V^2 \sim Q^2$.) With $D_1 = 3.0$ in., $D_2 = 1.0$ in., and $\rho = 1.94\text{ slug/ ft}^3$,

$$p_{1_g} = \frac{8}{\pi^2} \times 1.94\frac{\text{slug}}{\text{ft}^3} \times \frac{1}{(3)^4\text{in.}^4} \times Q^2[(3.0)^4 - 1]\frac{\text{lbf} \cdot \text{s}^2}{\text{slug} \cdot \text{ft}} \times 144\frac{\text{in.}^2}{\text{ft}^2}$$

$$p_{1_g} = 224\, Q^2 \frac{\text{lbf} \cdot \text{s}^2}{\text{in.}^2 \cdot \text{ft}^6}$$

With $Q = 0.7\text{ ft}^3/\text{s}$, then $p_{1_g} = 110\text{ lbf/in.}^2 \longleftarrow p_{1_g}$

This problem illustrates application of the Bernoulli equation to a flow where the restrictions of steady, incompressible, frictionless flow along a streamline are reasonable.

Control Volume Moving with Constant Velocity

In the preceding problems, which illustrate applications of the momentum equation to inertial control volumes, we have considered only stationary control volumes. Suppose we have a control volume moving at constant speed. We can set up two coordinate systems: XYZ, "absolute," or stationary (and therefore inertial), coordinates, and the xyz coordinates attached to the control volume (also inertial because the control volume is not accelerating with respect to XYZ).

Equation 4.10, which expresses system derivatives in terms of control volume variables, is valid for any motion of the control volume coordinate system xyz, provided that all velocities are measured *relative* to the control volume. To emphasize this point, we rewrite Eq. 4.10 as

$$\left.\frac{dN}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} \eta\,\rho\,dV\!\!\!/ + \int_{\text{CS}} \eta\,\rho\,\vec{V}_{xyz}\cdot d\vec{A} \tag{4.25}$$

Since all velocities must be measured relative to the control volume, in using this equation to obtain the momentum equation for an inertial control volume from the system formulation, we must set

$$N = \vec{P}_{xyz} \quad \text{and} \quad \eta = \vec{V}_{xyz}$$

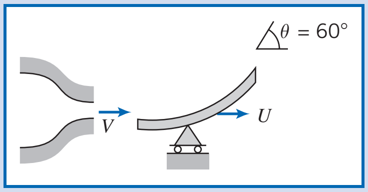
The control volume equation is then written as

$$\vec{F} = \vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t}\int_{\text{CV}} \vec{V}_{xyz}\,\rho\,dV\!\!\!/ + \int_{\text{CS}} \vec{V}_{xyz}\,\rho\vec{V}_{xyz}\cdot d\vec{A} \tag{4.26}$$

Equation 4.26 is the formulation of Newton's second law applied to any inertial control volume (stationary or moving with a constant velocity). It is identical to Eq. 4.17a except that we have included subscript xyz to emphasize that velocities must be measured relative to the control volume. (It is helpful to imagine that the velocities are those that would be seen by an observer moving with the control volume.) Example 4.10 illustrates the use of Eq. 4.26 for a control volume moving at constant velocity.

Example 4.10 VANE MOVING WITH CONSTANT VELOCITY

The sketch shows a vane with a turning angle of 60°. The vane moves at constant speed, $U = 10\,\text{m/s}$, and receives a jet of water that leaves a stationary nozzle with speed $V = 30\,\text{m/s}$. The nozzle has an exit area of $0.003\ \text{m}^2$. Determine the force components that act on the vane.

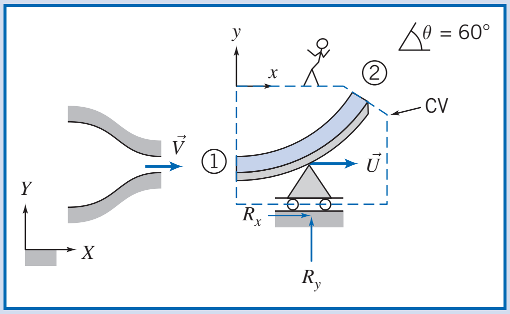


Given: Vane, with turning angle $\theta = 60°$, moves with constant velocity, $\vec{U} = 10\hat{i}\,\text{m/s}$. Water from a constant area nozzle, $A = 0.003\,\text{m}^2$, with velocity $\vec{V} = 30\hat{i}\,\text{m/s}$, flows over the vane as shown.

Find: Force components acting on the vane.

Solution: Select a control volume moving with the vane at constant velocity, $\vec{U}$, as shown by the dashed lines. R_x and R_y are the components of force required to maintain the velocity of the control volume at $10\hat{i}\,\text{m/s}$.

The control volume is inertial, since it is not accelerating ($U = \text{constant}$). Remember that all velocities must be measured relative to the control volume in applying the basic equations.



Governing equations:

$$\vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t}\int_{CV} \vec{V}_{xyz}\rho\, d\forall + \int_{CS} \vec{V}_{xyz}\, \rho\vec{V}_{xyz}\cdot d\vec{A}$$

$$\frac{\partial}{\partial t}\int_{CV} \rho\, d\forall + \int_{CS} \rho\vec{V}_{xyz}\cdot d\vec{A} = 0$$

Assumptions:
(1) Flow is steady relative to the vane.
(2) Magnitude of relative velocity along the vane is constant: $|\vec{V}_1| = |\vec{V}_2| = V - U$.
(3) Properties are uniform at sections 1 and 2.
(4) $F_{B_x} = 0$.
(5) Incompressible flow.

The x component of the momentum equation is

$$F_{S_x} + \cancelto{=0(4)}{F_{B_x}} = \cancelto{=0(1)}{\frac{\partial}{\partial t}}\int_{CV} u_{xyz}\,\rho\, d\forall + \int_{CS} u_{xyz}\,\rho\vec{V}_{xyz}\cdot d\vec{A}$$

There is no net pressure force, since p_{atm} acts on all sides of the CV. Thus

$$R_x = \int_{A_1} u(-\rho V dA) + \int_{A_2} u(\rho V dA) = +u_1(-\rho V_1 A_1) + u_2(\rho V_2 A_2)$$

(All velocities are measured relative to xyz.) From the continuity equation

$$\int_{A_1}(-\rho V dA) + \int_{A_2}(\rho V dA) = (-\rho V_1 A_1) + (\rho V_2 A_2) = 0$$

or

$$\rho V_1 A_1 = \rho V_2 A_2$$

Therefore,

$$R_x = (u_2 - u_1)(\rho V_1 A_1)$$

All velocities must be measured relative to the CV, so we note that

$$V_1 = V - U \qquad V_2 = V - U$$
$$u_1 = V - U \qquad u_2 = (V - U)\cos\theta$$

Substituting yields

$$R_x = [(V-U)\cos\theta - (V-U)](\rho(V-U)A_1) = (V-U)(\cos\theta - 1)\{\rho(V-U)A_1\}$$
$$= (30-10)\frac{\text{m}}{\text{s}} \times (0.50-1) \times \left(999\frac{\text{kg}}{\text{m}^3}(30-10)\frac{\text{m}}{\text{s}} \times 0.003\ \text{m}^2\right) \times \frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}}$$

$$R_x = -599\text{N}\ \{\text{to the left}\}$$

Writing the y component of the momentum equation, we obtain

$$F_{S_y} + F_{B_y} = \cancelto{=0(1)}{\frac{\partial}{\partial t}}\int_{CV} v_{xyz}\rho\, d\forall + \int_{CS} v_{xyz}\rho\,\vec{V}_{xyz}\cdot d\vec{A}$$

Denoting the mass of the CV as M gives

$$\begin{aligned} R_y - Mg &= \int_{\text{CS}} v\rho\vec{V}\cdot d\vec{A} = \int_{\text{A}_2} v\rho\vec{V}\cdot d\vec{A} \qquad \{v_1 = 0\} \\ &= \int_{\text{A}_2} v(\rho V dA) = v_2(\rho V_2 A_2) = v_2(\rho V_1 A_1) \\ &= (V - U)\sin\theta\{\rho(V-U)A_1\} \\ &= (30-10)\,\frac{\text{m}}{\text{s}} \times (0.866) \times \left((999)\,\frac{\text{kg}}{\text{n}^3}(30-10)\,\frac{\text{m}}{\text{s}} \times 0.003\text{m}^2\right) \times \frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}} \end{aligned}$$

{All velocities are measured relative to xyz.}

{Recall $\rho V_2 A_2 = \rho V_1 A_1$.}

$$R_y - Mg = 1.04\,\text{kN} \qquad \{\text{upward}\}$$

Thus the vertical force is

$$R_y = 1.04\,\text{kN} + Mg \qquad \{\text{upward}\}$$

Then the net force on the vane (neglecting the weight of the vane and water within the CV) is

$$\vec{R} = -0.599\hat{i} + 1.04\hat{j}\,\text{kN} \longleftarrow \vec{R}$$

This problem illustrates how to apply the momentum equation for a control volume in constant velocity motion by evaluating all velocities relative to the control volume.

4.5 Momentum Equation for Control Volume with Rectilinear Acceleration

For an inertial control volume (having no acceleration relative to a stationary frame of reference), the appropriate formulation of Newton's second law is given by Eq. 4.26,

$$\vec{F} = \vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t}\int_{\text{CV}} \vec{V}_{xyz}\,\rho\,d\forall + \int_{\text{CS}} \vec{V}_{xyz}\,\rho\vec{V}_{xyz}\cdot d\vec{A} \tag{4.26}$$

Not all control volumes are inertial; for example, a rocket must accelerate if it is to get off the ground. Since we are interested in analyzing control volumes that may accelerate relative to inertial coordinates, it is logical to ask whether Eq. 4.26 can be used for an accelerating control volume. To answer this question, let us briefly review the two major elements used in developing Eq. 4.26.

First, in relating the system derivatives to the control volume formulation (Eq. 4.25 or 4.10), the flow field, $\vec{V}(x, y, z, t)$, was specified relative to the control volume's coordinates x, y, and z. No restriction was placed on the motion of the xyz reference frame. Consequently, Eq. 4.25 (or Eq. 4.10) is valid at any instant for any arbitrary motion of the coordinates x, y, and z provided that all velocities in the equation are measured relative to the control volume.

Second, the system equation

$$\vec{F} = \left.\frac{d\vec{P}}{dt}\right)_{\text{system}} \tag{4.2a}$$

where the linear momentum of the system is given by

$$\vec{P}_{\text{system}} = \int_{M(\text{system})} \vec{V}\,dm = \int_{\mathcal{V}(\text{system})} \vec{V}\,\rho\,d\mathcal{V} \tag{4.2b}$$

is valid only for velocities measured relative to an inertial reference frame. Thus, if we denote the inertial reference frame by XYZ, then Newton's second law states that

$$\vec{F} = \left.\frac{d\vec{P}_{XYZ}}{dt}\right)_{\text{system}} \tag{4.27}$$

Since the time derivatives of $\vec{P}_{XYZ}$ and $\vec{P}_{xyz}$ are not equal when the control volume reference frame xyz is accelerating relative to the inertial reference frame, Eq. 4.26 is not valid for an accelerating control volume.

To develop the momentum equation for a linearly accelerating control volume, it is necessary to relate $\vec{P}_{XYZ}$ of the system to $\vec{P}_{xyz}$ of the system. The system derivative $d\vec{P}_{xyz}/dt$ can then be related to control volume variables through Eq. 4.25. We begin by writing Newton's second law for a system, remembering that the acceleration must be measured relative to an inertial reference frame that we have designated XYZ. We write

$$\vec{F} = \left.\frac{d\vec{P}_{XYZ}}{dt}\right)_{\text{system}} = \frac{d}{dt}\int_{M(\text{system})} \vec{V}_{XYZ}\,dm = \int_{M(\text{system})} \frac{d\vec{V}_{XYZ}}{dt}\,dm \tag{4.28}$$

The velocities with respect to the inertial (XYZ) and the control volume coordinates (xyz) are related by the relative-motion equation

$$\vec{V}_{XYZ} = \vec{V}_{xyz} + \vec{V}_{rf} \tag{4.29}$$

where $\vec{V}_{rf}$ is the velocity of the control volume coordinates xyz with respect to the "absolute" stationary coordinates XYZ.

Since we are assuming the motion of xyz is pure translation, without rotation, relative to inertial reference frame XYZ, then

$$\frac{d\vec{V}_{XYZ}}{dt} = \vec{a}_{XYZ} = \frac{d\vec{V}_{xyz}}{dt} + \frac{d\vec{V}_{rf}}{dt} = \vec{a}_{xyz} + \vec{a}_{rf} \tag{4.30}$$

where

$\vec{a}_{XYZ}$ is the rectilinear acceleration of the system relative to inertial reference frame XYZ,

$\vec{a}_{xyz}$ is the rectilinear acceleration of the system relative to noninertial reference frame xyz (i.e., relative to the control volume), and

$\vec{a}_{rf}$ is the rectilinear acceleration of noninertial reference frame xyz (i.e., of the control volume) relative to inertial frame XYZ.

Substituting from Eq. 4.30 into Eq. 4.28 gives

$$\vec{F} = \int_{M(\text{system})} \vec{a}_{rf}\,dm + \int_{M(\text{system})} \frac{d\vec{V}_{xyz}}{dt}\,dm$$

or

$$\vec{F} - \int_{M(\text{system})} \vec{a}_{rf}\,dm = \left.\frac{d\vec{P}_{xyz}}{dt}\right)_{\text{system}} \tag{4.31a}$$

where the linear momentum of the system is given by

$$\left.\vec{P}_{xyz}\right)_{\text{system}} = \int_{M(\text{system})} \vec{V}_{xyz}\,dm = \int_{\mathcal{V}(\text{system})} \vec{V}_{xyz}\,\rho\,d\mathcal{V} \tag{4.31b}$$

and the force, $\vec{F}$, includes all surface and body forces acting on the system.

To derive the control volume formulation of Newton's second law, we set

$$N = \vec{P}_{xyz} \qquad \text{and} \qquad \eta = \vec{V}_{xyz}$$

From Eq. 4.25, with this substitution, we obtain

$$\left.\frac{d\vec{P}_{xyz}}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} \vec{V}_{xyz}\rho\, d\maltese + \int_{\text{CS}} \vec{V}_{xyz}\rho\vec{V}_{xyz}\cdot d\vec{A} \tag{4.32}$$

Combining Eq. 4.31a (the linear momentum equation for the system) and Eq. 4.32 (the system–control volume conversion), and recognizing that at time t_0 the system and control volume coincide, Newton's second law for a control volume accelerating, without rotation, relative to an inertial reference frame is

$$\vec{F} - \int_{\text{CV}} \vec{a}_{rf}\, \rho\, d\maltese = \frac{\partial}{\partial t}\int_{\text{CV}} \vec{V}_{xyz}\, \rho\, d\maltese + \int_{\text{CS}} \vec{V}_{xyz}\, \rho\vec{V}_{xyz}\cdot d\vec{A}$$

Since $\vec{F} = \vec{F}_S + \vec{F}_B$, this equation becomes

$$\vec{F}_S + \vec{F}_B - \int_{\text{CV}} \vec{a}_{rf}\, \rho\, d\maltese = \frac{\partial}{\partial t}\int_{\text{CV}} \vec{V}_{xyz}\, \rho\, d\maltese + \int_{\text{CS}} \vec{V}_{xyz}\, \rho\vec{V}_{xyz}\cdot d\vec{A} \tag{4.33}$$

Comparing this momentum equation for a control volume with rectilinear acceleration to that for a nonaccelerating control volume, Eq. 4.26, we see that the only difference is the presence of one additional term in Eq. 4.33. When the control volume is not accelerating relative to inertial reference frame XYZ, then $\vec{a}_{rf} = 0$, and Eq. 4.33 reduces to Eq. 4.26.

The precautions concerning the use of Eq. 4.26 also apply to the use of Eq. 4.33. Before attempting to apply either equation, one must draw the boundaries of the control volume and label appropriate coordinate directions. For an accelerating control volume, one must label two coordinate systems: one (xyz) on the control volume and the other (XYZ) an inertial reference frame.

In Eq. 4.33, $\vec{F}_S$ represents all surface forces acting on the control volume. Since the mass within the control volume may vary with time, both the remaining terms on the left side of the equation may be functions of time. Furthermore, the acceleration, $\vec{a}_{rf}$, of the reference frame xyz relative to an inertial frame will in general be a function of time.

All velocities in Eq. 4.33 are measured relative to the control volume. The momentum flux, $\vec{V}_{xyz}\rho\vec{V}_{xyz}\cdot d\vec{A}$, through an element of the control surface area, $d\vec{A}$, is a vector. As we saw for the nonaccelerating control volume, the sign of the scalar product, $\rho\vec{V}_{xyz}\cdot d\vec{A}$, depends on the direction of the velocity vector, $\vec{V}_{xyz}$, relative to the area vector, $d\vec{A}$.

The momentum equation is a vector equation. As with all vector equations, it may be written as three scalar component equations. The scalar components of Eq. 4.33 are

$$F_{S_x} + F_{B_x} - \int_{\text{CV}} a_{rf_x}\, \rho d\maltese = \frac{\partial}{\partial t}\int_{\text{CV}} u_{xyz}\, \rho d\maltese + \int_{\text{CS}} u_{xyz}\, \rho\vec{V}_{xyz}\cdot d\vec{A} \tag{4.34a}$$

$$F_{S_y} + F_{B_y} - \int_{\text{CV}} a_{rf_y}\, \rho d\maltese = \frac{\partial}{\partial t}\int_{\text{CV}} v_{xyz}\, \rho d\maltese + \int_{\text{CS}} v_{xyz}\, \rho\vec{V}_{xyz}\cdot d\vec{A} \tag{4.34b}$$

$$F_{S_z} + F_{B_z} - \int_{\text{CV}} a_{rf_z}\, \rho d\maltese = \frac{\partial}{\partial t}\int_{\text{CV}} w_{xyz}\, \rho d\maltese + \int_{\text{CS}} w_{xyz}\, \rho\vec{V}_{xyz}\cdot d\vec{A} \tag{4.34c}$$

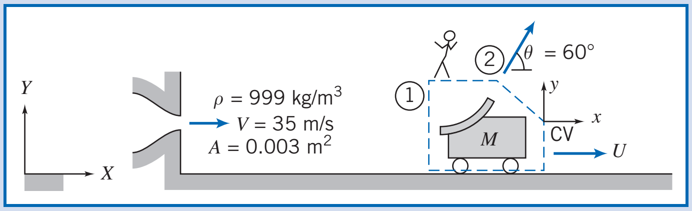
We will consider two applications of the linearly accelerating control volume: Example 4.11 will analyze an accelerating control volume in which the mass contained in the control volume is constant; Example 4.12 will analyze an accelerating control volume in which the mass contained varies with time.

Example 4.11 VANE MOVING WITH RECTILINEAR ACCELERATION

A vane, with turning angle $\theta = 60°$, is attached to a cart. The cart and vane, of mass $M = 75$ kg, roll on a level track. Friction and air resistance may be neglected. The vane receives a jet of water, which leaves a stationary nozzle horizontally at $V = 35$ m/s. The nozzle exit area is $A = 0.003\ \text{m}^2$. Determine the velocity of the cart as a function of time and plot the results.

Given: Vane and cart as sketched, with $M = 75$ kg.

Find: $U(t)$ and plot results.



Solution: Choose the control volume and coordinate systems shown for the analysis. Note that XY is a fixed frame, while frame xy moves with the cart. Apply the x component of the momentum equation.

Governing equation:

$$\underbrace{\cancel{F_{S_x}}}_{=0(1)} + \underbrace{\cancel{F_{B_x}}}_{=0(2)} - \int_{\text{CV}} a_{rf_x}\,\rho\,d\forall = \underbrace{\cancel{\frac{\partial}{\partial t}}\int_{\text{CV}} u_{xyz}\,\rho\,d\forall}_{\simeq 0(4)} + \int_{\text{CS}} u_{xyz}\,\rho\vec{V}_{xyz}\cdot d\vec{A}$$

Assumptions:
(1) $F_{S_x} = 0$, since no resistance is present.
(2) $F_{B_x} = 0$.
(3) Neglect the mass of water in contact with the vane compared to the cart mass.
(4) Neglect rate of change of momentum of liquid inside the CV.

$$\frac{\partial}{\partial t}\int_{\text{CV}} u_{xyz}\,\rho\,d\forall \simeq 0$$

(5) Uniform flow at sections 1 and 2.
(6) Speed of water stream is not slowed by friction on the vane, so $|\vec{V}_{xyz_1}| = |\vec{V}_{xyz_2}|$.
(7) $A_2 = A_1 = A$.

Then, dropping subscripts rf and xyz for clarity (but remembering that all velocities are measured relative to the moving coordinates of the control volume),

$$\begin{aligned}
-\int_{\text{CV}} a_x\,\rho\,d\forall &= u_1(-\rho V_1 A_1) + u_2(\rho V_2 A_2)\\
&= (V-U)\{-\rho(V-U)A\} + (V-U)\cos\theta\{\rho(V-U)A\}\\
&= -\rho(V-U)^2 A + \rho(V-U)^2 A\cos\theta
\end{aligned}$$

For the left side of this equation we have

$$-\int_{\text{CV}} a_x\,\rho\,d\forall = -a_x M_{CV} = -a_x M = -\frac{dU}{dt}M$$

so that

$$-M\frac{dU}{dt} = -\rho(V-U)^2 A + \rho(V-U)^2 A\cos\theta$$

or

$$M\,\frac{dU}{dt} = (1-\cos\theta)\rho(V-U)^2 A$$

Separating variables, we obtain

$$\frac{dU}{(V-U)^2} = \frac{(1-\cos\theta)\rho A}{M}dt = b\,dt \quad \text{where } b = \frac{(1-\cos\theta)\rho A}{M}$$

Note that since V = constant, $dU = -d(V-U)$. Integrating between limits $U = 0$ at $t = 0$, and $U = U$ at $t = t$,

$$\int_0^U \frac{dU}{(V-U)^2} = \int_0^U \frac{-d(V-U)}{(V-U)^2} = \frac{1}{(V-U)}\bigg]_0^U = \int_0^t b\,dt = bt$$

or

$$\frac{1}{(V-U)} - \frac{1}{V} = \frac{U}{V(V-U)} = bt$$

Solving for U, we obtain

$$\frac{U}{V} = \frac{Vbt}{1+Vbt}$$

Evaluating Vb gives

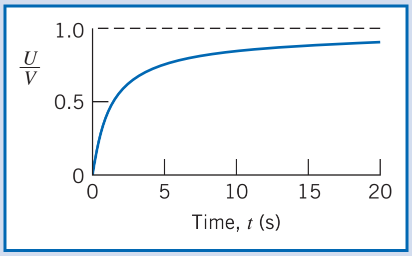
$$Vb = V\frac{(1-\cos\theta)\rho A}{M}$$

$$Vb = 35\,\frac{\text{m}}{\text{s}} \times \frac{(1-0.5)}{75\text{ kg}} \times 999\,\frac{\text{kg}}{\text{m}^3} \times 0.003\text{ m}^2 = 0.699\text{ s}^{-1}$$

Thus

$$\frac{U}{V} = \frac{0.699t}{1+0.699t} \longleftarrow \qquad (t \text{ in seconds}) \qquad U(t)$$

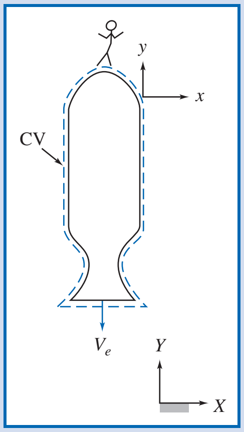
Plot:



The graph was generated from an *Excel* workbook. This workbook is interactive: It allows one to see the effect of different values of ρ, A, M, and θ on U/V against time t, and also to determine the time taken for the cart to reach, for example, 95% of jet speed.

Example 4.12 ROCKET DIRECTED VERTICALLY

A small rocket, with an initial mass of 400 kg, is to be launched vertically. Upon ignition the rocket consumes fuel at the rate of 5 kg/s and ejects gas at atmospheric pressure with a speed of 3500 m/s relative to the rocket. Determine the initial acceleration of the rocket and the rocket speed after 10 s, if air resistance is neglected.



Given: Small rocket accelerates vertically from rest.
Initial mass, $M_0 = 400\,\text{kg}$.
Air resistance may be neglected.
Rate of fuel consumption, $\dot{m}_e = 5\,\text{kg/s}$.
Exhaust velocity, $V_e = 3500\,\text{m/s}$, relative to rocket, leaving at atmospheric pressure.

Find: (a) Initial acceleration of the rocket.
(b) Rocket velocity after 10 s.

Solution:
Choose a control volume as shown by dashed lines. Because the control volume is accelerating, define inertial coordinate system XY and coordinate system xy attached to the CV. Apply the y component of the momentum equation.

Governing equation: $F_{S_y} + F_{B_y} - \int_{\text{CV}} a_{rf_y}\,\rho\,d\mathcal{V} = \dfrac{\partial}{\partial t}\int_{\text{CV}} v_{xyz}\,\rho\,d\mathcal{V} + \int_{\text{CS}} v_{xyz}\,\rho\vec{V}_{xyz}\cdot d\vec{A}$

Assumptions: (1) Atmospheric pressure acts on all surfaces of the CV; since air resistance is neglected, $F_{S_y} = 0$.
(2) Gravity is the only body force; g is constant.
(3) Flow leaving the rocket is uniform, and V_e is constant.

Under these assumptions the momentum equation reduces to

$$\underset{\text{A}}{F_{B_y}} - \underset{\text{B}}{\int_{\text{CV}} a_{rf_y}\,\rho\,d\mathcal{V}} = \underset{\text{C}}{\frac{\partial}{\partial t}\int_{\text{CV}} v_{xyz}\,\rho\,d\mathcal{V}} + \underset{\text{D}}{\int_{\text{CS}} v_{xyz}\,\rho\vec{V}_{xyz}\cdot d\vec{A}} \tag{1}$$

Let us look at the equation term by term:

A $\quad F_{B_y} = -\int_{\text{CV}} g\,\rho\,d\mathcal{V} = -g\int_{\text{CV}} \rho\,d\mathcal{V} = -gM_{\text{CV}} \qquad$ {since g is constant}

The mass of the CV will be a function of time because mass is leaving the CV at rate $\dot{m}_e$. To determine M_{CV} as a function of time, we use the conservation of mass equation

$$\frac{\partial}{\partial t}\int_{\text{CV}} \rho\,d\mathcal{V} + \int_{\text{CS}} \rho\vec{V}\cdot d\vec{A} = 0$$

Then

$$\frac{\partial}{\partial t}\int_{\text{CV}} \rho\,d\mathcal{V} = -\int_{\text{CS}} \rho\vec{V}\cdot d\vec{A} = -\int_{\text{CS}} (\rho V_{xyz}dA) = -\dot{m}_e$$

The minus sign indicates that the mass of the CV is decreasing with time. Since the mass of the CV is only a function of time, we can write

$$\frac{dM_{\text{CV}}}{dt} = -\dot{m}_e$$

To find the mass of the CV at any time, t, we integrate

$$\int_{M_0}^{M} dM_{\text{CV}} = -\int_0^t \dot{m}_e\,dt \qquad \text{where at } t = 0,\ M_{\text{CV}} = M_0, \text{ and at } t = t,\ M_{\text{CV}} = M$$

Then, $M - M_0 = -\dot{m}_e t$, or $M = M_0 - \dot{m}_e t$.

Substituting the expression for M into term A, we obtain

$$F_{B_y} = -\int_{\text{CV}} g\,\rho\,d\mathcal{V} = -gM_{\text{CV}} = -g(M_0 - \dot{m}_e t)$$

B $\quad -\int_{\text{CV}} a_{rf_y}\,\rho\,d\mathcal{V}$

The acceleration, a_{rf_y}, of the CV is that seen by an observer in the XY coordinate system. Thus a_{rf_y} is not a function of the coordinates xyz, and

$$-\int_{\mathrm{CV}} a_{rf_y}\,\rho\,d\forall = -a_{rf_y}\int_{\mathrm{CV}}\rho\,d\forall = -a_{rf_y}\,M_{\mathrm{CV}} = -a_{rf_y}(\,M_0 - \dot{m}_e t)$$

Ⓒ $\dfrac{\partial}{\partial t}\displaystyle\int_{\mathrm{CV}} v_{xyz}\,\rho\,d\forall$

This is the time rate of change of the y momentum of the fluid in the control volume measured relative to the control volume.

Even though the y momentum of the fluid inside the CV, measured relative to the CV, is a large number, it does not change appreciably with time. To see this, we must recognize that:

(1) The unburned fuel and the rocket structure have zero momentum relative to the rocket.
(2) The velocity of the gas at the nozzle exit remains constant with time as does the velocity at various points in the nozzle.

Consequently, it is reasonable to assume that

$$\frac{\partial}{\partial t}\int_{\mathrm{CV}} v_{xyz}\,\rho\,d\forall \approx 0$$

Ⓓ $\displaystyle\int_{\mathrm{CS}} v_{xyz}\,\rho\vec{V}_{xyz}\cdot d\vec{A} = \int_{\mathrm{CS}} v_{xyz}(\rho V_{xyz}\,dA) = -V_e\int_{\mathrm{CS}}(\rho V_{xyz}\,dA)$

The velocity v_{xyz} (relative to the control volume) is $-V_e$ (it is in the negative y direction), and is a constant, so was taken outside the integral. The remaining integral is simply the mass flow rate at the exit (positive because flow is out of the control volume),

$$\int_{\mathrm{CS}}(\rho V_{xyz} dA) = \dot{m}_e$$

and so

$$\int_{\mathrm{CS}} v_{xyz}\,\rho\vec{V}_{xyz}\cdot d\vec{A} = -V_e\dot{m}_e$$

Substituting terms Ⓐ through Ⓓ into Eq. 1, we obtain

$$-g(M_0 - \dot{m}_e t) - a_{rf_y}(M_0 - \dot{m}_e t) = -V_e\dot{m}_e$$

or

$$a_{rf_y} = \frac{V_e\dot{m}_e}{M_0 - \dot{m}_e t} - g \tag{2}$$

At time $t = 0$,

$$a_{rf_y})_{t=0} = \frac{V_e\dot{m}_e}{M_0} - g = 3500\,\frac{\mathrm{m}}{\mathrm{s}}\times 5\,\frac{\mathrm{kg}}{\mathrm{s}}\times\frac{1}{400\,\mathrm{kg}} - 9.81\,\frac{\mathrm{m}}{\mathrm{s}^2}$$

$$a_{rf_y})_{t=0} = 33.9\,\mathrm{m/s^2} \longleftarrow \underline{a_{rf_y})_{t=0}}$$

The acceleration of the CV is by definition

$$a_{rf_y} = \frac{dV_{\mathrm{CV}}}{dt}$$

Substituting from Eq. 2,

$$\frac{dV_{CV}}{dt} = \frac{V_e \dot{m}_e}{M_0 - \dot{m}_e t} - g$$

Separating variables and integrating gives

$$V_{CV} = \int_0^{V_{CV}} dV_{CV} = \int_0^t \frac{V_e \dot{m}_e dt}{M_0 - \dot{m}_e t} - \int_0^t g dt = -V_e \ln\left[\frac{M_0 - \dot{m}_e t}{M_0}\right] - gt$$

At $t = 10\,\text{s}$,

$$V_{CV} = -3500\,\frac{\text{m}}{\text{s}} \times \ln\left[\frac{350\,\text{kg}}{400\,\text{kg}}\right] - 9.81\,\frac{\text{m}}{\text{s}^2} \times 10\,\text{s}$$

$$V_{CV} = 369\,\text{m/s} \longleftarrow V_{CV})_{t\,=\,10\,\text{s}}$$

The velocity-time graph is shown in an *Excel* workbook. This workbook is interactive: It allows one to see the effect of different values of M_o, V_e, and $\dot{m}_e$ on V_{CV} versus time t. Also, the time at which the rocket attains a given speed, e.g., 2000 m/s, can be determined.

Momentum Equation For Control Volume With Arbitrary Acceleration (on the Web) 4.6

The Angular-Momentum Principle 4.7*

Our next task is to derive a control volume form of the angular-momentum principle. There are two obvious approaches we can use to express the angular-momentum principle: We can use an inertial (fixed) XYZ control volume; we can also use a rotating xyz control volume. For each approach we will: start with the principle in its system form (Eq. 4.3a), then write the system angular momentum in terms of XYZ or xyz coordinates, and finally use Eq. 4.10 (or its slightly different form, Eq. 4.25) to convert from a system to a control volume formulation. To verify that these two approaches are equivalent, we will use each approach to solve the same problem, in Examples 4.14 and 4.15 (on the Web), respectively.

There are two reasons for the material of this section: We wish to develop a control volume equation for each of the basic physical laws of Section 4.2; and we will need the results for use in Chapter 10, where we discuss rotating machinery.

Equation for Fixed Control Volume

The angular-momentum principle for a system in an inertial frame is

$$\vec{T} = \left.\frac{d\vec{H}}{dt}\right)_{\text{system}} \tag{4.3a}$$

where $\vec{T}$ = total torque exerted on the system by its surroundings, and
$\vec{H}$ = angular momentum of the system.

*This section may be omitted without loss of continuity in the text material.

$$\vec{H} = \int_{M(\text{system})} \vec{r} \times \vec{V}\, dm = \int_{\forall(\text{system})} \vec{r} \times \vec{V}\, \rho\, d\forall \tag{4.3b}$$

All quantities in the system equation must be formulated with respect to an inertial reference frame. Reference frames at rest, or translating with constant linear velocity, are inertial, and Eq. 4.3b can be used directly to develop the control volume form of the angular-momentum principle.

The position vector, $\vec{r}$, locates each mass or volume element of the system with respect to the coordinate system. The torque, $\vec{T}$, applied to a system may be written

$$\vec{T} = \vec{r} \times \vec{F}_s + \int_{M(\text{system})} \vec{r} \times \vec{g}\, dm + \vec{T}_{\text{shaft}} \tag{4.3c}$$

where $\vec{F}_s$ is the surface force exerted on the system.

The relation between the system and fixed control volume formulations is

$$\left.\frac{dN}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} \eta\, \rho\, d\forall + \int_{\text{CS}} \eta\, \rho \vec{V} \cdot d\vec{A} \tag{4.10}$$

where

$$N_{\text{system}} = \int_{M(\text{system})} \eta\, dm$$

If we set $N = \vec{H}$, then $\eta = \vec{r} \times \vec{V}$, and

$$\left.\frac{d\vec{H}}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} \vec{r} \times \vec{V}\, \rho\, d\forall + \int_{\text{CS}} \vec{r} \times \vec{V}\, \rho \vec{V} \cdot d\vec{A} \tag{4.45}$$

Combining Eqs. 4.3a, 4.3c, and 4.45, we obtain

$$\vec{r} \times \vec{F}_s + \int_{M(\text{system})} \vec{r} \times \vec{g}\, dm + \vec{T}_{\text{shaft}} = \frac{\partial}{\partial t}\int_{\text{CV}} \vec{r} \times \vec{V}\, \rho\, d\forall + \int_{\text{CS}} \vec{r} \times \vec{V}\, \rho \vec{V} \cdot d\vec{A}$$

Since the system and control volume coincide at time t_0,

$$\vec{T} = \vec{T}_{\text{CV}}$$

and

$$\vec{r} \times \vec{F}_s + \int_{\text{CV}} \vec{r} \times \vec{g}\, \rho\, d\forall + \vec{T}_{\text{shaft}} = \frac{\partial}{\partial t}\int_{\text{CV}} \vec{r} \times \vec{V}\, \rho\, d\forall + \int_{\text{CS}} \vec{r} \times \vec{V}\, \rho \vec{V} \cdot d\vec{A} \tag{4.46}$$

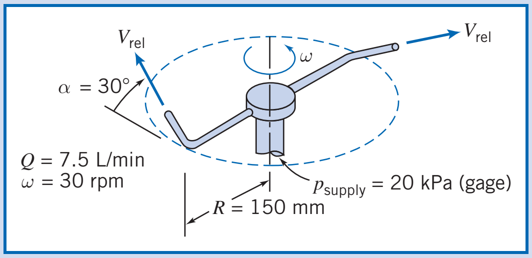
Equation 4.46 is a general formulation of the angular-momentum principle for an inertial control volume. The left side of the equation is an expression for all the torques that act on the control volume. Terms on the right express the rate of change of angular momentum within the control volume and the net rate of flux of angular momentum from the control volume. All velocities in Eq. 4.46 are measured relative to the fixed control volume.

For analysis of rotating machinery, Eq. 4.46 is often used in scalar form by considering only the component directed along the axis of rotation. This application is illustrated in Chapter 10.

The application of Eq. 4.46 to the analysis of a simple lawn sprinkler is illustrated in Example 4.14. This same problem is considered in Example 4.15 (on the Web) using the angular-momentum principle expressed in terms of a *rotating* control volume.

Example 4.14 LAWN SPRINKLER: ANALYSIS USING FIXED CONTROL VOLUME

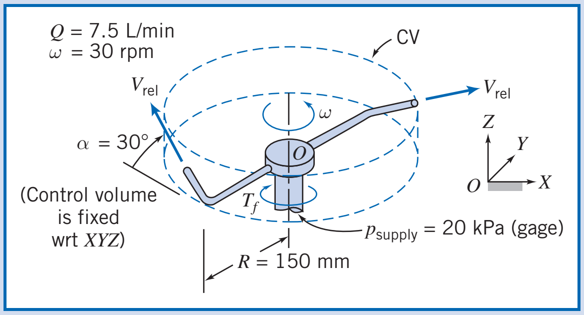
A small lawn sprinkler is shown in the sketch at right. At an inlet gage pressure of 20 kPa, the total volume flow rate of water through the sprinkler is 7.5 liters per minute and it rotates at 30 rpm. The diameter of each jet is 4 mm. Calculate the jet speed relative to each sprinkler nozzle. Evaluate the friction torque at the sprinkler pivot.



Given: Small lawn sprinkler as shown.

Find: (a) Jet speed relative to each nozzle.
(b) Friction torque at pivot.

Solution: Apply continuity and angular momentum equations using fixed control volume enclosing sprinkler arms.



Governing equations:

$$\overbrace{\cancel{\frac{\partial}{\partial t}\int_{CV} \rho\, d\mathcal{V}}}^{=0(1)} + \int_{CS} \rho \vec{V}\cdot d\vec{A} = 0$$

$$\vec{r}\times\vec{F}_s + \int_{CV} \vec{r}\times\vec{g}\,\rho\, d\mathcal{V} + \vec{T}_{\text{shaft}} = \frac{\partial}{\partial t}\int_{CV} \vec{r}\times\vec{V}\rho\, d\mathcal{V} + \int_{CS} \vec{r}\times\vec{V}\,\rho\vec{V}\cdot d\vec{A} \tag{1}$$

where all velocities are measured relative to the inertial coordinates XYZ.

Assumptions: (1) Incompressible flow.
(2) Uniform flow at each section.
(3) $\vec{\omega}$ = constant.

From continuity, the jet speed relative to the nozzle is given by

$$V_{\text{rel}} = \frac{Q}{2A_{\text{jet}}} = \frac{Q}{2}\frac{4}{\pi D_{\text{jet}}^2}$$

$$= \frac{1}{2}\times 7.5\frac{L}{\text{min}}\times\frac{4}{\pi}\frac{1}{(4)^2\,\text{mm}^2}\times\frac{\text{m}^3}{1000\,\text{L}}\times 10^6\frac{\text{mm}^2}{\text{m}^2}\times\frac{\text{min}}{60\,\text{s}}$$

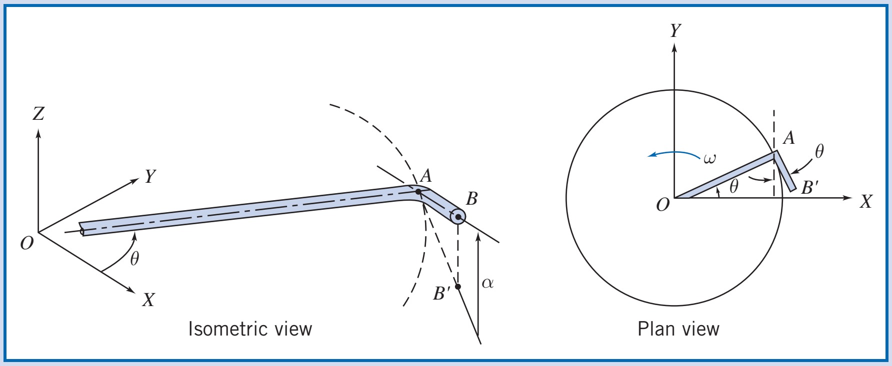
$$V_{\text{rel}} = 4.97\,\text{m/s} \longleftarrow V_{\text{rel}}$$

Consider terms in the angular momentum equation separately. Since atmospheric pressure acts on the entire control surface, and the pressure force at the inlet causes no moment about O, $\vec{r}\times\vec{F}_s = 0$. The moments of the body (i.e., gravity) forces in the two arms are equal and opposite and hence the second term on the left side of the equation is zero. The only external torque acting on the CV is friction in the pivot. It opposes the motion, so

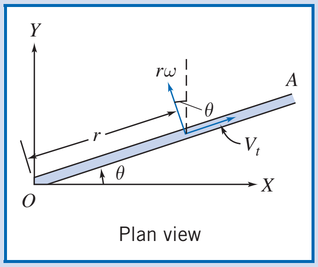
$$\vec{T}_{\text{shaft}} = -T_f\hat{K} \tag{2}$$

Our next task is to determine the two angular momentum terms on the right side of Eq. 1. Consider the unsteady term: This is the rate of change of angular momentum in the control volume. It is clear that although the position $\vec{r}$ and velocity $\vec{V}$ of fluid particles are functions of time in XYZ coordinates, because the sprinkler rotates at constant speed the control volume angular momentum *is* constant in XYZ coordinates, so this term is zero; however, as an exercise in manipulating vector quantities, let us derive this result. Before we can evaluate the control volume integral, we need to develop expressions for the instantaneous position vector, $\vec{r}$, and velocity vector, $\vec{V}$ (measured relative to the fixed coordinate system XYZ) of each element of fluid in the control volume. OA lies in the XY plane; AB is inclined at angle α to the XY plane; point B' is the projection of point B on the XY plane.

We assume that the length, L, of the tip AB is small compared with the length, R, of the horizontal arm OA. Consequently we neglect the angular momentum of the fluid in the tips compared with the angular momentum in the horizontal arms.



Isometric view   Plan view



Plan view

Consider flow in the horizontal tube OA of length R. Denote the radial distance from O by r. At any point in the tube the fluid velocity relative to fixed coordinates XYZ is the sum of the velocity relative to the tube $\vec{V}_t$ and the tangential velocity $\vec{\omega} \times \vec{r}$. Thus

$$\vec{V} = \hat{I}(V_t \cos\theta - r\omega \sin\theta) + \hat{J}(V_t \sin\theta + r\omega\cos\theta)$$

(Note that θ is a function of time.) The position vector is

$$\vec{r} = \hat{I} r\cos\theta + \hat{J} r\sin\theta$$

and

$$\vec{r} \times \vec{V} = \hat{K}(r^2\omega\cos^2\theta + r^2\omega\sin^2\theta) = \hat{K} r^2\omega$$

Then

$$\int_{V\!\!\!\!-_{OA}} \vec{r}\times\vec{V}\,\rho\, dV\!\!\!\!- = \int_O^R \hat{K} r^2 \omega\rho A\, dr = \hat{K}\frac{R^3\omega}{3}\rho A$$

and

$$\frac{\partial}{\partial t}\int_{V\!\!\!\!-_{OA}} \vec{r}\times\vec{V}\rho\, dV\!\!\!\!- = \frac{\partial}{\partial t}\left[\hat{K}\frac{R^3\omega}{3}\rho A\right] = 0 \qquad (3)$$

where A is the cross-sectional area of the horizontal tube. Identical results are obtained for the other horizontal tube in the control volume. We have confirmed our insight that the angular momentum within the control volume does not change with time.

Now we need to evaluate the second term on the right, the flux of momentum across the control surface. There are three surfaces through which we have mass and therefore momentum flux: the supply line (for which $\vec{r}\times\vec{V} = 0$) because $\vec{r} = 0$ and the two nozzles. Consider the nozzle at the end of branch OAB. For $L \ll R$, we have

$$\vec{r}_{\text{jet}} = \vec{r}_B \approx \vec{r}\big|_{r=R} = (\hat{I} r\cos\theta + \hat{J} r\sin\theta)\big|_{r=R} = \hat{I}R\cos\theta + \hat{J}R\sin\theta$$

and for the instantaneous jet velocity $\vec{V}_j$ we have

$$\vec{V}_j = \vec{V}_{\text{rel}} + \vec{V}_{\text{tip}} = \hat{I}V_{\text{rel}}\cos\alpha\sin\theta - \hat{J}V_{\text{rel}}\cos\alpha\cos\theta + \hat{K}V_{\text{rel}}\sin\alpha - \hat{I}\omega R\sin\theta + \hat{J}\omega R\cos\theta$$

$$\vec{V}_j = \hat{I}(V_{\text{rel}}\cos\alpha - \omega R)\sin\theta - \hat{J}(V_{\text{rel}}\cos\alpha - \omega R)\cos\theta + \hat{K}V_{\text{rel}}\sin\alpha$$

$$\vec{r}_B \times \vec{V}_j = \hat{I}RV_{\text{rel}}\sin\alpha\sin\theta - \hat{J}RV_{\text{rel}}\sin\alpha\cos\theta - \hat{K}R(V_{\text{rel}}\cos\alpha - \omega R)(\sin^2\theta + \cos^2\theta)$$

$$\vec{r}_B \times \vec{V}_j = \hat{I}RV_{\text{rel}}\sin\alpha\sin\theta - \hat{J}RV_{\text{rel}}\sin\alpha\cos\theta - \hat{K}R(V_{\text{rel}}\cos\alpha - \omega R)$$

The flux integral evaluated for flow crossing the control surface at location B is then

$$\int_{CS} \vec{r} \times \vec{V}_j \, \rho \vec{V} \cdot d\vec{A} = \left[\hat{I} R V_{\text{rel}} \sin \alpha \sin \theta - \hat{J} R V_{\text{rel}} \sin \alpha \cos \theta - \hat{K} R (V_{\text{rel}} \cos \alpha - \omega R)\right] \rho \frac{Q}{2}$$

The velocity and radius vectors for flow in the left arm must be described in terms of the same unit vectors used for the right arm. In the left arm the $\hat{I}$ and $\hat{J}$ components of the cross product are of opposite sign, since $\sin(\theta + \pi) = -\sin(\theta)$ and $\cos(\theta + \pi) = -\cos(\theta)$. Thus for the complete CV,

$$\int_{CS} \vec{r} \times \vec{V}_j \, \rho \vec{V} \cdot d\vec{A} = -\hat{K} R (V_{\text{rel}} \cos \alpha - \omega R) \rho Q \qquad (4)$$

Substituting terms (2), (3), and (4) into Eq. 1, we obtain

$$-T_f \hat{K} = -\hat{K} R (V_{\text{rel}} \cos \alpha - \omega R) \rho Q$$

or

$$T_f = R (V_{\text{rel}} \cos \alpha - \omega R) \rho Q$$

This expression indicates that when the sprinkler runs at constant speed the friction torque at the sprinkler pivot just balances the torque generated by the angular momentum of the two jets.

From the data given,

$$\omega R = 30 \frac{\text{rev}}{\text{min}} \times 150 \text{ mm} \times 2\pi \frac{\text{rad}}{\text{rev}} \times \frac{\text{min}}{60 \text{ s}} \times \frac{\text{m}}{1000 \text{ mm}} = 0.471 \text{ m/s}$$

Substituting gives

$$T_f = 150 \text{ mm} \times \left(4.97 \frac{\text{m}}{\text{s}} \times \cos 30^\circ - 0.471 \frac{\text{m}}{\text{s}}\right) 999 \frac{\text{kg}}{\text{m}^3} \times 7.5 \frac{\text{L}}{\text{min}} \times \frac{\text{m}^3}{1000 \text{ L}} \times \frac{\text{min}}{60 \text{ s}} \times \frac{\text{N} \cdot \text{s}^2}{\text{kg} \cdot \text{m}} \times \frac{\text{m}}{1000 \text{ mm}}$$

$$T_f = 0.0718 \text{ N} \cdot \text{m} \xleftarrow{\hspace{6em} T_f}$$

This problem illustrates use of the angular momentum principle for an inertial control volume. Note that in this example the fluid particle position vector $\vec{r}$ and velocity vector $\vec{V}$ are time-dependent (through θ) in XYZ coordinates. This problem will be solved again using a noninertial (rotating) xyz coordinate system in Example 4.15 (on the Web).

Equation for Rotating Control Volume (on the Web)

The First Law of Thermodynamics 4.8

The first law of thermodynamics is a statement of conservation of energy. Recall that the system formulation of the first law was

$$\dot{Q} - \dot{W} = \left.\frac{dE}{dt}\right)_{\text{system}} \qquad (4.4a)$$

where the total energy of the system is given by

$$E_{\text{system}} = \int_{M(\text{system})} e \, dm = \int_{\mathcal{V}(\text{system})} e \, \rho \, d\mathcal{V} \qquad (4.4b)$$

and

$$e = u + \frac{V^2}{2} + gz$$

In Eq. 4.4a, the rate of heat transfer, $\dot{Q}$, is positive when heat is added *to* the system from the surroundings; the rate of work, $\dot{W}$, is positive when work is done *by* the system on its surroundings. (Note that some texts use the opposite notation for work.)

To derive the control volume formulation of the first law of thermodynamics, we set

$$N = E \qquad \text{and} \qquad \eta = e$$

in Eq. 4.10 and obtain

$$\left.\frac{dE}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} e\,\rho\, d\forall + \int_{\text{CS}} e\,\rho \vec{V} \cdot d\vec{A} \tag{4.53}$$

Since the system and the control volume coincide at t_0,

$$[\dot{Q} - \dot{W}]_{\text{system}} = [\dot{Q} - \dot{W}]_{\text{control volume}}$$

In light of this, Eqs. 4.4a and 4.53 yield the control volume form of the first law of thermodynamics,

$$\dot{Q} - \dot{W} = \frac{\partial}{\partial t}\int_{\text{CV}} e\,\rho\, d\forall + \int_{\text{CS}} e\,\rho \vec{V} \cdot d\vec{A} \tag{4.54}$$

where

$$e = u + \frac{V^2}{2} + gz$$

Note that for steady flow the first term on the right side of Eq. 4.54 is zero.

Is Eq. 4.54 the form of the first law used in thermodynamics? Even for steady flow, Eq. 4.54 is not quite the same form used in applying the first law to control volume problems. To obtain a formulation suitable and convenient for problem solutions, let us take a closer look at the work term, $\dot{W}$.

Rate of Work Done by a Control Volume

The term $\dot{W}$ in Eq. 4.54 has a positive numerical value when work is done by the control volume on the surroundings. The rate of work done *on* the control volume is of opposite sign to the work done *by* the control volume.

The rate of work done by the control volume is conveniently subdivided into four classifications,

$$\dot{W} = \dot{W}_s + \dot{W}_{\text{normal}} + \dot{W}_{\text{shear}} + \dot{W}_{\text{other}}$$

Let us consider these separately:

1. *Shaft Work*

We shall designate shaft work W_s and hence the rate of work transferred out through the control surface by shaft work is designated $\dot{W}_s$. Examples of shaft work are the work produced by the steam turbine (positive shaft work) of a power plant, and the work input required to run the compressor of a refrigerator (negative shaft work).

2. Work Done by Normal Stresses at the Control Surface

Recall that work requires a force to act through a distance. Thus, when a force, $\vec{F}$, acts through an infinitesimal displacement, $d\vec{s}$, the work done is given by

$$\delta W = \vec{F} \cdot d\vec{s}$$

To obtain the rate at which work is done by the force, divide by the time increment, Δt, and take the limit as $\Delta t \to 0$. Thus the rate of work done by the force, $\vec{F}$, is

$$\dot{W} = \lim_{\Delta t \to 0} \frac{\delta W}{\Delta t} = \lim_{\Delta t \to 0} \frac{\vec{F} \cdot d\vec{s}}{\Delta t} \qquad \text{or} \qquad \dot{W} = \vec{F} \cdot \vec{V}$$

We can use this to compute the rate of work done by the normal and shear stresses. Consider the segment of control surface shown in Fig. 4.6. For an element of area $d\vec{A}$ we can write an expression for the normal stress force $d\vec{F}_{\text{normal}}$: It will be given by the normal stress σ_{nn} multiplied by the vector area element $d\vec{A}$ (normal to the control surface).

Hence the rate of work done on the area element is

$$d\vec{F}_{\text{normal}} \cdot \vec{V} = \sigma_{nn}\, d\vec{A} \cdot \vec{V}$$

Since the work out across the boundaries of the control volume is the negative of the work done on the control volume, the total rate of work out of the control volume due to normal stresses is

$$\dot{W}_{\text{normal}} = -\int_{\text{CS}} \sigma_{nn}\, d\vec{A} \cdot \vec{V} = -\int_{\text{CS}} \sigma_{nn} \vec{V} \cdot d\vec{A}$$

3. Work Done by Shear Stresses at the Control Surface

Just as work is done by the normal stresses at the boundaries of the control volume, so may work be done by the shear stresses.

As shown in Fig. 4.6, the shear force acting on an element of area of the control surface is given by

$$d\vec{F}_{\text{shear}} = \vec{\tau}\, dA$$

where the shear stress vector, $\vec{\tau}$, is the shear stress acting in some direction in the plane of dA.

The rate of work done on the entire control surface by shear stresses is given by

$$\int_{\text{CS}} \vec{\tau}\, dA \cdot \vec{V} = \int_{\text{CS}} \vec{\tau} \cdot \vec{V} dA$$

Since the work out across the boundaries of the control volume is the negative of the work done on the control volume, the rate of work out of the control volume due to shear stresses is given by

$$\dot{W}_{\text{shear}} = -\int_{\text{CS}} \vec{\tau} \cdot \vec{V}\, dA$$

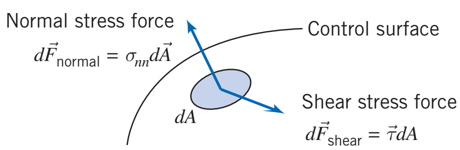


Fig. 4.6 Normal and shear stress forces.

This integral is better expressed as three terms

$$\begin{aligned}\dot{W}_{\text{shear}} &= -\int_{\text{CS}} \vec{\tau} \cdot \vec{V}\, dA \\ &= -\int_{A(\text{shafts})} \vec{\tau} \cdot \vec{V}\, dA - \int_{A(\text{solid surface})} \vec{\tau} \cdot \vec{V}\, dA - \int_{A(\text{ports})} \vec{\tau} \cdot \vec{V}\, dA\end{aligned}$$

We have already accounted for the first term, since we included $\dot{W}_s$ previously. At solid surfaces, $\vec{V} = 0$, so the second term is zero (for a fixed control volume). Thus,

$$\dot{W}_{\text{shear}} = -\int_{A(\text{ports})} \vec{\tau} \cdot \vec{V}\, dA$$

This last term can be made zero by proper choice of control surfaces. If we choose a control surface that cuts across each port perpendicular to the flow, then $d\vec{A}$ is parallel to $\vec{V}$. Since $\vec{\tau}$ is in the plane of dA, $\vec{\tau}$ is perpendicular to $\vec{V}$. Thus, for a control surface perpendicular to $\vec{V}$,

$$\vec{\tau} \cdot \vec{V} = 0 \qquad \text{and} \qquad \dot{W}_{\text{shear}} = 0$$

4. Other Work

Electrical energy could be added to the control volume. Also electromagnetic energy, e.g., in radar or laser beams, could be absorbed. In most problems, such contributions will be absent, but we should note them in our general formulation.

With all of the terms in $\dot{W}$ evaluated, we obtain

$$\dot{W} = \dot{W}_s - \int_{\text{CS}} \sigma_{nn}\, \vec{V} \cdot d\vec{A} + \dot{W}_{\text{shear}} + \dot{W}_{\text{other}} \tag{4.55}$$

Control Volume Equation

Substituting the expression for $\dot{W}$ from Eq. 4.55 into Eq. 4.54 gives

$$\dot{Q} - \dot{W}_s + \int_{\text{CS}} \sigma_{nn}\, \vec{V} \cdot d\vec{A} - \dot{W}_{\text{shear}} - \dot{W}_{\text{other}} = \frac{\partial}{\partial t}\int_{\text{CV}} e\, \rho\, d\forall + \int_{\text{CS}} e\, \rho \vec{V} \cdot d\vec{A}$$

Rearranging this equation, we obtain

$$\dot{Q} - \dot{W}_s - \dot{W}_{\text{shear}} - \dot{W}_{\text{other}} = \frac{\partial}{\partial t}\int_{\text{CV}} e\, \rho\, d\forall + \int_{\text{CS}} e\, \rho \vec{V} \cdot d\vec{A} - \int_{\text{CS}} \sigma_{nn}\, \vec{V} \cdot d\vec{A}$$

Since $\rho = 1/v$, where v is *specific volume*, then

$$\int_{\text{CS}} \sigma_{nn}\, \vec{V} \cdot d\vec{A} = \int_{\text{CS}} \sigma_{nn}\, v\, \rho \vec{V} \cdot d\vec{A}$$

Hence

$$\dot{Q} - \dot{W}_s - \dot{W}_{\text{shear}} - \dot{W}_{\text{other}} = \frac{\partial}{\partial t}\int_{\text{CV}} e\, \rho\, d\forall + \int_{\text{CS}} (e - \sigma_{nn} v)\, \rho \vec{V} \cdot d\vec{A}$$

Viscous effects can make the normal stress, σ_{nn}, different from the negative of the thermodynamic pressure, $-p$. However, for most flows of common engineering interest, $\sigma_{nn} \simeq -p$. Then

$$\dot{Q} - \dot{W}_s - \dot{W}_{\text{shear}} - \dot{W}_{\text{other}} = \frac{\partial}{\partial t}\int_{\text{CV}} e\, \rho\, d\forall + \int_{\text{CS}} (e + pv)\, \rho \vec{V} \cdot d\vec{A}$$

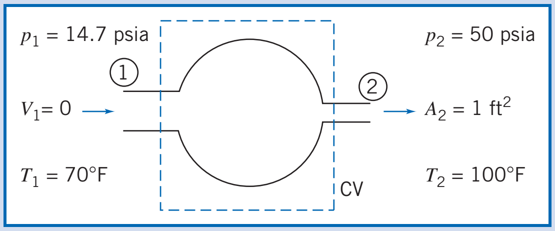
Finally, substituting $e = u + V^2/2 + gz$ into the last term, we obtain the familiar form of the first law for a control volume,

$$\dot{Q} - \dot{W}_s - \dot{W}_{\text{shear}} - \dot{W}_{\text{other}} = \frac{\partial}{\partial t}\int_{\text{CV}} e\,\rho\,d\forall + \int_{\text{CS}}\left(u + pv + \frac{V^2}{2} + gz\right)\rho\vec{V}\cdot d\vec{A} \quad (4.56)$$

Each work term in Eq. 4.56 represents the rate of work done by the control volume on the surroundings. Note that in thermodynamics, for convenience, the combination $u + pv$ (the fluid internal energy plus what is often called the "flow work") is usually replaced with enthalpy, $h \equiv u + pv$ (this is one of the reasons h was invented).

Example 4.16 COMPRESSOR: FIRST LAW ANALYSIS

Air at 14.7 psia, 70°F, enters a compressor with negligible velocity and is discharged at 50 psia, 100°F through a pipe with 1 ft^2 area. The flow rate is 20 lbm/s. The power input to the compressor is 600 hp. Determine the rate of heat transfer.



Given: Air enters a compressor at 1 and leaves at 2 with conditions as shown. The air flow rate is 20 lbm/s and the power input to the compressor is 600 hp.

Find: Rate of heat transfer.

Solution:

Governing equations:

$$\overset{=0(1)}{\cancel{\frac{\partial}{\partial t}\int_{\text{CV}}\rho\,d\forall}} + \int_{\text{CS}}\rho\vec{V}\cdot d\vec{A} = 0$$

$$\dot{Q} - \dot{W}_s - \overset{=0(4)}{\cancel{\dot{W}_{\text{shear}}}} = \overset{=0(1)}{\cancel{\frac{\partial}{\partial t}\int_{\text{CV}} e\,\rho\,d\forall}} + \int_{\text{CS}}\left(u + pv + \frac{V^2}{2} + gz\right)\rho\vec{V}\cdot d\vec{A}$$

Assumptions:
(1) Steady flow.
(2) Properties uniform over inlet and outlet sections.
(3) Treat air as an ideal gas, $p = \rho RT$.
(4) Area of CV at 1 and 2 perpendicular to velocity, thus $\dot{W}_{\text{shear}} = 0$.
(5) $z_1 = z_2$.
(6) Inlet kinetic energy is negligible.

Under the assumptions listed, the first law becomes

$$\dot{Q} - \dot{W}_s = \int_{\text{CV}}\left(u + pv + \frac{V^2}{2} + gz\right)\rho\vec{V}\cdot d\vec{A}$$

$$\dot{Q} - \dot{W}_s = \int_{\text{CS}}\left(h + \frac{V^2}{2} + gz\right)\rho\vec{V}\cdot d\vec{A}$$

or

$$\dot{Q} = \dot{W}_s + \int_{\text{CS}}\left(h + \frac{V^2}{2} + gz\right)\rho\vec{V}\cdot d\vec{A}$$

For uniform properties, assumption (2), we can write

$$\dot{Q} = \dot{W}_s + \left(h_1 + \overset{\approx 0(6)}{\cancel{\frac{V_1^2}{2}}} + gz_1\right)(-\rho_1 V_1 A_1) + \left(h_2 + \frac{V_2^2}{2} + gz_2\right)(\rho_2 V_2 A_2)$$

For steady flow, from conservation of mass,

$$\int_{\text{CS}} \rho \vec{V} \cdot d\vec{A} = 0$$

Therefore, $-(\rho_1 V_1 A_1) + (\rho_2 V_2 A_2) = 0$, or $\rho_1 V_1 A_1 = \rho_2 V_2 A_2 = \dot{m}$. Hence we can write

$$\dot{Q} = \dot{W}_s + \dot{m}\left[(h_2 - h_1) + \frac{V_2^2}{2} + g\overset{=0(5)}{\cancel{(z_2 - z_1)}}\right]$$

Assume that air behaves as an ideal gas with constant c_p. Then $h_2 - h_1 = c_p(T_2 - T_1)$, and

$$\dot{Q} = \dot{W}_s + \dot{m}\left[c_p(T_2 - T_1) + \frac{V_2^2}{2}\right]$$

From continuity $V_2 = \dot{m}/\rho_2 A_2$. Since $p_2 = \rho_2 R T_2$,

$$V_2 = \frac{\dot{m}}{A_2}\frac{RT_2}{p_2} = 20\,\frac{\text{lbm}}{s} \times \frac{1}{1\,\text{ft}^2} \times 53.3\,\frac{\text{ft}\cdot\text{lbf}}{\text{lbm}\cdot{}^\circ\text{R}} \times 560^\circ\text{R} \times \frac{\text{in.}^2}{50\,\text{lbf}} \times \frac{\text{ft}^2}{144\,\text{in.}^2}$$

$$V_2 = 82.9\,\text{ft/s}$$

Note that power input is *to* the CV, so $\dot{W}_s = -600\,\text{hp}$, and

$$\dot{Q} = \dot{W}_s + \dot{m}c_p\,(T_2 - T_1) + \dot{m}\,\frac{V_2^2}{2}$$

$$\dot{Q} = -600\,\text{hp} \times 550\,\frac{\text{ft}\cdot\text{lbf}}{\text{hp}\cdot\text{s}} \times \frac{\text{Btu}}{778\,\text{ft}\cdot\text{lbf}} + 20\,\frac{\text{lbm}}{s} \times 0.24\,\frac{\text{Btu}}{\text{lbm}\cdot{}^\circ\,\text{R}} \times 30^\circ\text{R}$$

$$+ 20\,\frac{\text{lbm}}{\text{s}} \times \frac{(82.9)^2}{2}\frac{\text{ft}^2}{\text{s}^2} \times \frac{\text{slug}}{32.2\,\text{lbm}} \times \frac{\text{Btu}}{778\,\text{ft}\cdot\text{lbf}} \times \frac{\text{lbf}\cdot\text{s}^2}{\text{slug}\cdot\text{ft}}$$

$$\dot{Q} = -277\,\text{Btu/s} \longleftarrow \{\text{heat rejection}\}\dot{Q}$$

This problem illustrates use of the first law of thermodynamics for a control volume. It is also an example of the care that must be taken with unit conversions for mass, energy, and power.

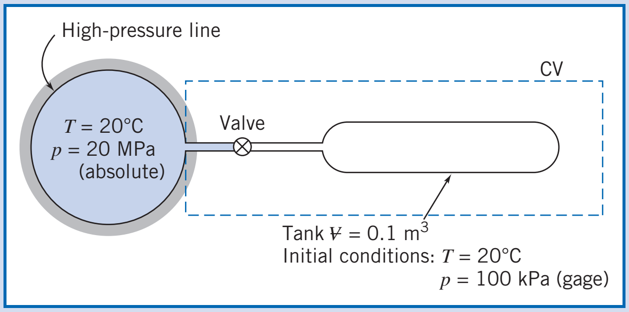
Example 4.17 TANK FILLING: FIRST LAW ANALYSIS

A tank of 0.1 m³ volume is connected to a high-pressure air line; both line and tank are initially at a uniform temperature of 20°C. The initial tank gage pressure is 100 kPa. The absolute line pressure is 2.0 MPa; the line is large enough so that its temperature and pressure may be assumed constant. The tank temperature is monitored by a fast-response thermocouple. At the instant after the valve is opened, the tank temperature rises at the rate of 0.05°C/s. Determine the instantaneous flow rate of air into the tank if heat transfer is neglected.

Given: Air supply pipe and tank as shown. At $t = 0^+$, $\partial T/\partial t = 0.05^\circ\text{C/s}$.

Find: $\dot{m}$ at $t = 0^+$.

Solution:
Choose CV shown, apply energy equation.



Governing equation:

$$\cancelto{0(1)}{\dot{Q}} - \cancelto{0(2)}{\dot{W}_s} - \cancelto{0(3)}{\dot{W}_{\text{shear}}} - \cancelto{0(4)}{\dot{W}_{\text{other}}} = \frac{\partial}{\partial t}\int_{\text{CV}} e\,\rho\, d\forall + \int_{\text{CS}} (e + pv)\rho \vec{V} \cdot d\vec{A}$$

$$e = u + \cancelto{\simeq 0(5)}{\frac{V^2}{2}} + \cancelto{\simeq 0(6)}{gz}$$

Assumptions:
(1) $\dot{Q} = 0$ (given).
(2) $\dot{W}_s = 0$.
(3) $\dot{W}_{\text{shear}} = 0$.
(4) $\dot{W}_{\text{other}} = 0$.
(5) Velocities in line and tank are small.
(6) Neglect potential energy.
(7) Uniform flow at tank inlet.
(8) Properties uniform in tank.
(9) Ideal gas, $p = \rho R T$, $du = c_v dT$.

Then

$$\frac{\partial}{\partial t}\int_{\text{CV}} u_{\text{tank}}\rho\, d\forall + (u + pv)|_{\text{line}}(-\rho V A) = 0$$

This expresses the fact that the gain in energy in the tank is due to influx of fluid energy (in the form of enthalpy $h = u + pv$) from the line. We are interested in the initial instant, when T is uniform at 20°C, so $u_{\text{tank}} = u_{\text{line}} = u$, the internal energy at T; also, $pv_{\text{line}} = RT_{\text{line}} = RT$, and

$$\frac{\partial}{\partial t}\int_{\text{CV}} u\,\rho\, d\forall + (u + RT)(-\rho V A) = 0$$

Since tank properties are uniform, $\partial/\partial t$ may be replaced by d/dt, and

$$\frac{d}{dt}(uM) = (u + RT)\dot{m}$$

(where M is the instantaneous mass in the tank and $\dot{m} = \rho V A$ is the mass flow rate), or

$$u\frac{dM}{dt} + M\frac{du}{dt} = u\dot{m} + RT\dot{m} \qquad (1)$$

The term dM/dt may be evaluated from continuity:

Governing equation:

$$\frac{\partial}{\partial t}\int_{CV}\rho d\forall + \int_{CS}\rho \vec{V}\cdot d\vec{A} = 0$$

$$\frac{dM}{dt} + (-\rho VA) = 0 \quad \text{or} \quad \frac{dM}{dt} = \dot{m}$$

Substituting in Eq. 1 gives

$$u\dot{m} + Mc_v\frac{dT}{dt} = u\dot{m} + RT\dot{m}$$

or

$$\dot{m} = \frac{Mc_v(dT/dt)}{RT} = \frac{\rho\forall c_v(dT/dt)}{RT} \tag{2}$$

But at $t = 0$, $p_{\text{tank}} = 100$ kPa (gage), and

$$\rho = \rho_{\text{tank}} = \frac{p_{\text{tank}}}{RT} = (1.00 + 1.01)10^5\frac{\text{N}}{\text{m}^2}\times\frac{\text{kg}\cdot\text{K}}{287\,\text{N}\cdot\text{m}}\times\frac{1}{293\,\text{K}}$$
$$= 2.39\,\text{kg/m}^3$$

Substituting into Eq. 2, we obtain

$$\dot{m} = 2.39\frac{\text{kg}}{\text{m}^3}\times 0.1\,\text{m}^3\times 717\frac{\text{N}\cdot\text{m}}{\text{kg}\cdot\text{K}}\times 0.05\frac{\text{K}}{\text{s}}$$
$$\times\frac{\text{kg}\cdot\text{K}}{287\,\text{N}\cdot\text{m}}\times\frac{1}{293\,\text{K}}\times 1000\frac{\text{g}}{\text{kg}}$$

$$\dot{m} = 0.102\,\text{g/s} \xleftarrow{\qquad\qquad\qquad\dot{m}\qquad\qquad\qquad}$$

This problem illustrates use of the first law of thermodynamics for a control volume. It is also an example of the care that must be taken with unit conversions for mass, energy, and power.

4.9 The Second Law of Thermodynamics

Recall that the system formulation of the second law is

$$\left.\frac{dS}{dt}\right)_{\text{system}} \geq \frac{1}{T}\dot{Q} \tag{4.5a}$$

where the total entropy of the system is given by

$$S_{\text{system}} = \int_{M(\text{system})} s\,dm = \int_{\forall(\text{system})} s\,\rho\,d\forall \tag{4.5b}$$

To derive the control volume formulation of the second law of thermodynamics, we set

$$N = S \qquad \text{and} \qquad \eta = s$$

in Eq. 4.10 and obtain

$$\left.\frac{dS}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} s\,\rho\, d\forall + \int_{\text{CS}} s\,\rho \vec{V}\cdot d\vec{A} \tag{4.57}$$

The system and the control volume coincide at t_0; thus in Eq. 4.5a,

$$\left.\frac{1}{T}\dot{Q}\right)_{\text{system}} = \left.\frac{1}{T}\dot{Q}\right)_{\text{CV}} = \int_{\text{CS}} \frac{1}{T}\left(\frac{\dot{Q}}{A}\right)dA$$

In light of this, Eqs. 4.5a and 4.57 yield the control volume formulation of the second law of thermodynamics

$$\frac{\partial}{\partial t}\int_{\text{CV}} s\,\rho\, d\forall + \int_{\text{CS}} s\,\rho\,\vec{V}\cdot d\vec{A} \geq \int_{\text{CS}} \frac{1}{T}\left(\frac{\dot{Q}}{A}\right)dA \tag{4.58}$$

In Eq. 4.58, the factor $(\dot{Q}/A)$ represents the heat flux per unit area into the control volume through the area element dA. To evaluate the term

$$\int_{\text{CS}} \frac{1}{T}\left(\frac{\dot{Q}}{A}\right)dA$$

both the local heat flux, $(\dot{Q}/A)$, and local temperature, T, must be known for each area element of the control surface.

4.10 Summary and Useful Equations

In this chapter we wrote the basic laws for a system: mass conservation (or continuity), Newton's second law, the angular-momentum equation, the first law of thermodynamics, and the second law of thermodynamics. We then developed an equation (sometimes called the Reynolds Transport Theorem) for relating system formulations to control volume formulations. Using this we derived control volume forms of:

- ✓ The mass conservation equation (sometimes called the continuity equation).
- ✓ Newton's second law (in other words, a momentum equation) for:
  - ○ An inertial control volume.
  - ○ A control volume with rectilinear acceleration.
  - ○ A control volume with arbitrary acceleration (on the Web).
- ✓ The angular-momentum equation for:*
  - ○ A fixed control volume.
  - ○ A rotating control volume (on the Web).
- ✓ The first law of thermodynamics (or energy equation).
- ✓ The second law of thermodynamics.

We discussed the physical meaning of each term appearing in these control volume equations, and used the equations for the solution of a variety of flow problems. In particular, we used a differential control volume* to derive a famous equation in fluid mechanics—the Bernoulli equation—and while doing so learned about the restrictions on its use in solving problems.

*These topics apply to a section that may be omitted without loss of continuity in the text material

Note: Most of the Useful Equations in the table below have a number of constraints or limitations—*be sure to refer to their page numbers for details*!

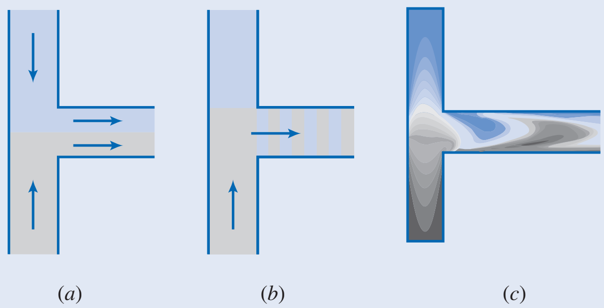
Useful Equations

Continuity (mass conservation), incompressible fluid:	$\int_{CS} \vec{V} \cdot d\vec{A} = 0$	**(4.13a)**	Page 105
Continuity (mass conservation), incompressible fluid, uniform flow:	$\sum_{CS} \vec{V} \cdot \vec{A} = 0$	**(4.13b)**	Page 105
Continuity (mass conservation), steady flow:	$\int_{CS} \rho\vec{V} \cdot d\vec{A} = 0$	**(4.15a)**	Page 106
Continuity (mass conservation), steady flow, uniform flow:	$\sum_{CS} \rho\vec{V} \cdot \vec{A} = 0$	**(4.15b)**	Page 106
Momentum (Newton's second law):	$\vec{F} = \vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t}\int_{CV} \vec{V}\,\rho\, dV\!\!\!/ + \int_{CS} \vec{V}\,\rho\vec{V} \cdot d\vec{A}$	**(4.17a)**	Page 111
Momentum (Newton's second law), uniform flow:	$\vec{F} = \vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t}\int_{CV} \vec{V}\,\rho\, dV\!\!\!/ + \sum_{CS} \vec{V}\rho\vec{V} \cdot \vec{A}$	**(4.17b)**	Page 111
Momentum (Newton's second law), scalar components:	$F_x = F_{S_x} + F_{B_x} = \frac{\partial}{\partial t}\int_{CV} u\,\rho\, dV\!\!\!/ + \int_{CS} u\,\rho\vec{V} \cdot d\vec{A}$	**(4.18a)**	Page 112
	$F_y = F_{S_y} + F_{B_y} = \frac{\partial}{\partial t}\int_{CV} v\,\rho\, dV\!\!\!/ + \int_{CS} v\,\rho\vec{V} \cdot d\vec{A}$	**(4.18b)**	
	$F_z = F_{S_z} + F_{B_z} = \frac{\partial}{\partial t}\int_{CV} w\,\rho\, dV\!\!\!/ + \int_{CS} w\,\rho\vec{V} \cdot d\vec{A}$	**(4.18c)**	
Momentum (Newton's second law), uniform flow, scalar components:	$F_x = F_{S_x} + F_{B_x} = \frac{\partial}{\partial t}\int_{CV} u\,\rho\, dV\!\!\!/ + \sum_{CS} u\,\rho\vec{V} \cdot \vec{A}$	**(4.18d)**	Page 112
	$F_y = F_{S_y} + F_{B_y} = \frac{\partial}{\partial t}\int_{CV} v\,\rho\, dV\!\!\!/ + \sum_{CS} v\,\rho\vec{V} \cdot \vec{A}$	**(4.18e)**	
	$F_z = F_{S_z} + F_{B_z} = \frac{\partial}{\partial t}\int_{CV} w\,\rho\, dV\!\!\!/ + \sum_{CS} w\,\rho\vec{V} \cdot \vec{A}$	**(4.18f)**	
Bernoulli equation (steady, incompressible, frictionless, flow along a streamline):	$\frac{p}{\rho} + \frac{V^2}{2} + gz = \text{constant}$	**(4.24)**	Page 124
Momentum (Newton's second law), inertial control volume (stationary or constant speed):	$\vec{F} = \vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t}\int_{CV} \vec{V}_{xyz}\,\rho\, dV\!\!\!/ + \int_{CS} \vec{V}_{xyz}\,\rho\vec{V}_{xyz} \cdot d\vec{A}$	**(4.26)**	Page 126
Momentum (Newton's second law), rectilinear acceleration of control volume:	$\vec{F}_S + \vec{F}_B - \int_{CV} \vec{a}_{rf}\,\rho\, dV\!\!\!/ = \frac{\partial}{\partial t}\int_{CV} \vec{V}_{xyz}\,\rho\, dV\!\!\!/ \int_{CS} \vec{V}_{xyz}\,\rho\vec{V}_{xyz} \cdot d\vec{A}$	**(4.33)**	Page 130
Angular-momentum principle:	$\vec{r} \times \vec{F}_s + \int_{CV} \vec{r} \times \vec{g}\,\rho\, dV\!\!\!/ + \vec{T}_{\text{shaft}} = \frac{\partial}{\partial t}\int_{CV} \vec{r} \times \vec{V}\,\rho\, dV\!\!\!/ + \int_{CS} \vec{r} \times \vec{V}\,\rho\vec{V} \cdot d\vec{A}$	**(4.46)**	Page 136

First law of thermodynamics:	$\dot{Q} - \dot{W}_s - \dot{W}_{\text{shear}} - \dot{W}_{\text{other}} = \frac{\partial}{\partial t}\int_{\text{CV}} e\,\rho\, dV\!\!\!\!- + \int_{\text{CS}} \left(u + pv + \frac{V^2}{2} + gz\right) \rho \vec{V} \cdot d\vec{A}$	**(4.56)**	Page 143
Second law of thermodynamics:	$\frac{\partial}{\partial t}\int_{\text{CV}} s\,\rho\, dV\!\!\!\!- + \int_{\text{CS}} s\,\rho\,\vec{V} \cdot d\vec{A} \geq \int_{\text{CS}} \frac{1}{T}\left(\frac{\dot{Q}}{A}\right) dA$	**(4.58)**	Page 147

Case Study

"Lab-on-a-Chip"



Mixing two fluids in a "lab-on-a-chip."

An exciting new area in fluid mechanics is microfluidics, applied to microelectromechanical systems (MEMS—the technology of very small devices, generally ranging in size from a micrometer to a millimeter). In particular, a lot of research is being done in "lab-on-a-chip" technology, which has many applications. An example of this is in medicine, with devices for use in the immediate point-of-care diagnosis of diseases, such as real-time detection of bacteria, viruses, and cancers in the human body. In the area of security, there are devices that continuously sample and test air or water samples for biochemical toxins and other dangerous pathogens such as those in always-on early warning systems.

Because of the extremely small geometry, flows in such devices will be very low Reynolds numbers and therefore laminar; surface tension effects will also be significant. In many common applications (for example, typical water pipes and air conditioning ducts), laminar flow would be desirable, but the flow is turbulent—it costs more to pump a turbulent as opposed to a laminar flow. In certain applications, turbulence is desirable instead because it acts as a mixing mechanism. If you couldn't generate turbulence in your coffee cup, it would take a lot of stirring before the cream and coffee were sufficiently blended; if your blood flow never became turbulent, you would not get sufficient oxygen to your organs and muscles! In the lab-on-a-chip, turbulent flow is usually desirable because the goal in these devices is often to mix minute amounts of two or more fluids.

How do we mix fluids in such devices that are inherently laminar? We could use complex geometries, or relatively long channels (relying on molecular diffusion), or some kind of MEM device with paddles. Research by professors Goullet, Glasgow, and Aubry at the New Jersey Institute of Technology instead suggests pulsing the two fluids. Part *a* of the figure shows a schematic of two fluids flowing at a constant rate (about 25 nL/s, average velocity less than 2 mm/s, in ducts about 200 μm wide) and meeting in a T junction. The two fluids do not mix because of the strongly laminar nature of the flow. Part *b* of the figure shows a schematic of an instant of a pulsed flow, and part *c* shows an instant computed using a computational fluid dynamics (CFD) model of the same flow. In this case, the interface between the two fluid samples is shown to stretch and fold, leading to good nonturbulent mixing within 2 mm downstream of the confluence (after about 1 s of contact). Such a compact mixing device would be ideal for many of the applications mentioned above.

Problems

Basic Laws for a System

4.1 A mass of 5 lbm is released when it is just in contact with a spring of stiffness 25 lbf/ft that is attached to the ground. What is the maximum spring compression? Compare this to the deflection if the mass was just resting on the compressed spring. What would be the maximum spring compression if the mass was released from a distance of 5 ft above the top of the spring?

4.2 An ice-cube tray containing 250 mL of freshwater at 15°C is placed in a freezer at −5°C. Determine the change in internal energy (kJ) and entropy (kJ/K) of the water when it has frozen.

4.3 A small steel ball of radius $r = 1$ mm is placed on top of a horizontal pipe of outside radius $R = 50$ mm and begins to roll under the influence of gravity. Rolling resistance and air resistance are negligible. As the speed of the ball increases, it

eventually leaves the surface of the pipe and becomes a projectile. Determine the speed and location at which the ball loses contact with the pipe.

4.4 A fully loaded Boeing 777-200 jet transport aircraft weighs 325,000 kg. The pilot brings the 2 engines to full takeoff thrust of 450 kN each before releasing the brakes. Neglecting aerodynamic and rolling resistance, estimate the minimum runway length and time needed to reach a takeoff speed of 225 kph. Assume engine thrust remains constant during ground roll.

4.5 A police investigation of tire marks showed that a car traveling along a straight and level street had skidded to a stop for a total distance of 200 ft after the brakes were applied. The coefficient of friction between tires and pavement is estimated to be μ = 0.7. What was the probable minimum speed (mph) of the car when the brakes were applied? How long did the car skid?

4.6 A high school experiment consists of a block of mass 2 kg sliding across a surface (coefficient of friction μ = 0.6). If it is given an initial velocity of 5 m/s, how far will it slide, and how long will it take to come to rest? The surface is now roughened a little, so with the same initial speed it travels a distance of 2 m. What is the new coefficient of friction, and how long does it now slide?

4.7 A car traveling at 30 mph encounters a curve in the road. The radius of the road curve is 100 ft. Find the maximum speeds (mph) before losing traction, if the coefficient of friction on a dry road is μ_{dry} = 0.7 and on a wet road is μ_{wet} = 0.3.

4.8 Air at 20°C and an absolute pressure of 1 atm is compressed adiabatically in a piston-cylinder device, without friction, to an absolute pressure of 4 atm in a piston-cylinder device. Find the work done (MJ).

4.9 In an experiment with a can of soda, it took 2 hr to cool from an initial temperature of 80°F to 45°F in a 35°F refrigerator. If the can is now taken from the refrigerator and placed in a room at 72°F, how long will the can take to reach 60°F? You may assume that for both processes the heat transfer is modeled by $\dot{Q} \approx k(T - T_{\text{amb}})$, where T is the can temperature, T_{amb} is the ambient temperature, and k is a heat transfer coefficient.

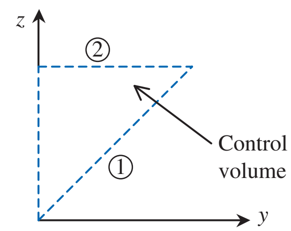
4.10 A block of copper of mass 5 kg is heated to 90°C and then plunged into an insulated container containing 4 L of water at 10°C. Find the final temperature of the system. For copper, the specific heat is 385 J/kg·K, and for water the specific heat is 4186 J/kg·K.

4.11 The average rate of heat loss from a person to the surroundings when not actively working is about 85 W. Suppose that in an auditorium with volume of approximately 3.5×10^5 m^3, containing 6000 people, the ventilation system fails. How much does the internal energy of the air in the auditorium increase during the first 15 min after the ventilation system fails? Considering the auditorium and people as a system, and assuming no heat transfer to the surroundings, how much does the internal energy of the system change? How do you account for the fact that the temperature of the air increases? Estimate the rate of temperature rise under these conditions.

Conservation of Mass

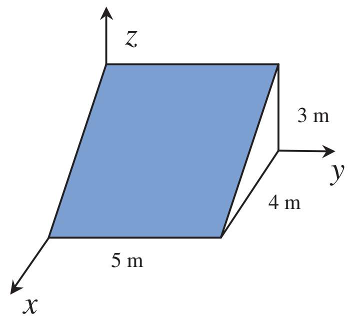
4.12 The velocity field in the region shown is given by $\vec{V} = (a\hat{j} + by\hat{k})$ where $a = 10$ m/s and $b = 5\,\text{s}^{-1}$. For the 1 m × 1 m triangular control volume (depth $w = 1$ m perpendicular to the diagram), an element of area ① may be represented by $d\vec{A}_1 = wdz\hat{j} - wdy\hat{k}$ and an element of area ② by $d\vec{A}_2 = -wdy\hat{k}$.

(a) Find an expression for $\vec{V} \cdot dA_1$.
(b) Evaluate $\int_{A_1} \vec{V} \cdot dA_1$.
(c) Find an expression for $\vec{V} \cdot dA_2$.
(d) Find an expression for $\vec{V}(\vec{V} \cdot dA_2)$.
(e) Evaluate $\int_{A_2} \vec{V}(\vec{V} \cdot dA_2)$.



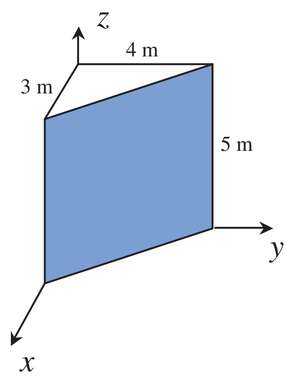
P4.12

4.13 The shaded area shown is in a flow where the velocity field is given by $\vec{V} = ax\hat{i} + by\hat{j}$; $a = b = 1\,\text{s}^{-1}$, and the coordinates are measured in meters. Evaluate the volume flow rate and the momentum flux through the shaded area ($\rho = 1$ kg/m^3).



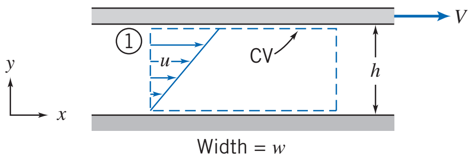
P4.13

4.14 The area shown shaded is in a flow where the velocity field is given by $\vec{V} = ax\hat{i} + by\hat{j} + c\hat{k}$; $a = b = 2\,\text{s}^{-1}$ and $c = 1$ m/s. Write a vector expression for an element of the shaded area. Evaluate the integrals $\int_A \vec{V} \cdot dA$ and $\int_A \vec{V}(\vec{V} \cdot d\vec{A})$ over the shaded area.



P4.14

4.15 Obtain expressions for the volume flow rate and the momentum flux through cross section ① of the control volume shown in the diagram.



P4.15

4.16 For the flow of Problem 4.15, obtain an expression for the kinetic energy flux, $\int (V^2/2)\rho\vec{V}\cdot d\vec{A}$, through cross section ① of the control volume shown.

4.17 The velocity distribution for laminar flow in a long circular tube of radius R is given by the one-dimensional expression,

$$\vec{V} = u\hat{i} = u_{\max}\left[1 - \left(\frac{r}{R}\right)^2\right]\hat{i}$$

For this profile obtain expressions for the volume flow rate and the momentum flux through a section normal to the pipe axis.

4.18 For the flow of Problem 4.17, obtain an expression for the kinetic energy flux, $\int (V^2/2)\rho\vec{V}\cdot d\vec{A}$, through a section normal to the pipe axis.

4.19 A shower head fed by a ¾-in. ID water pipe consists of 50 nozzles of 1/32-in. ID. Assuming a flow rate of 2.2 gpm, what is the exit velocity (ft/s) of each jet of water? What is the average velocity (ft/s) in the pipe?

4.20 A farmer is spraying a liquid through 10 nozzles, ⅛-in. ID, at an average exit velocity of 10 ft/s. What is the average velocity in the 1-in. ID head feeder? What is the system flow rate, in gpm?

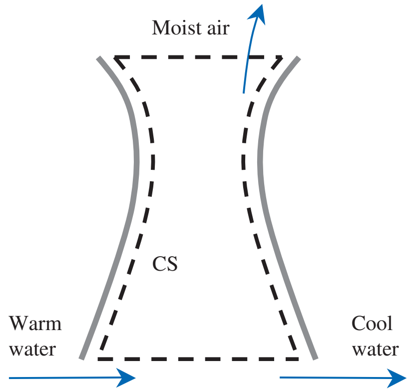
4.21 A cylindrical holding water tank has a 3 m ID and a height of 3 m. There is one inlet of diameter 10 cm, an exit of diameter 8 cm, and a drain. The tank is initially empty when the inlet pump is turned on, producing an average inlet velocity of 5 m/s. When the level in the tank reaches 0.7 m, the exit pump turns on, causing flow out of the exit; the exit average velocity is 3 m/s. When the water level reaches 2 m the drain is opened such that the level remains at 2 m. Find (a) the time at which the exit pump is switched on, (b) the time at which the drain is opened, and (c) the flow rate into the drain (m^3/min).

4.22 A university laboratory that generates 15 m^3/s of air flow at design condition wishes to build a wind tunnel with variable speeds. It is proposed to build the tunnel with a sequence of three circular test sections: section 1 will have a diameter of 1.5 m, section 2 a diameter of 1 m, and section 3 a diameter such that the average speed is 75 m/s.

(a) What will be the speeds in sections 1 and 2?

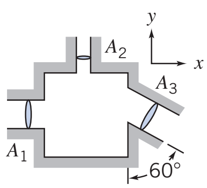
(b) What must the diameter of section 3 be to attain the desired speed at design condition?

4.23 A wet cooling tower cools warm water by spraying it into a forced dry-air flow. Some of the water evaporates in this air and is carried out of the tower into the atmosphere; the evaporation cools the remaining water droplets, which are collected at the exit pipe (6 in. ID) of the tower. Measurements indicate the warm water mass flow rate is 250,000 lb/hr, and the cool water (70°F) flows at an average speed of 5 ft/s in the exit pipe. The moist air density is 0.065 lb/ft^3. Find (a) the volume flow rate (ft^3/s) and mass flow rate (lb/hr) of the cool water, (b) the mass flow rate (lb/hr) of the moist air, and (c) the mass flow rate (lb/hr) of the dry air. *Hint*: Google "density of moist air" for information on relating moist and dry air densities!

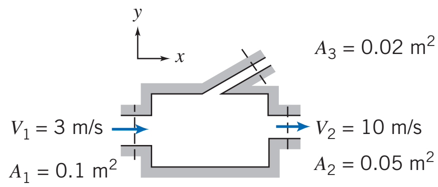


P4.23

4.24 Fluid with 65 lbm/ft^3 density is flowing steadily through the rectangular box shown. Given $A_1 = 0.5$ ft^2, $A_2 = 0.1$ ft^2, $A_3 = 0.6$ ft^2, $\vec{V}_1 = 10\hat{i}$ ft/s, and $\vec{V}_2 = 20\hat{j}$ ft/s, determine velocity $\vec{V}_3$.



P4.24



P4.25

4.25 Consider steady, incompressible flow through the device shown. Determine the magnitude and direction of the volume flow rate through port 3.

4.26 A rice farmer needs to fill her 150 m × 400 m field with water to a depth of 7.5 cm in 1 hr. How many 37.5-cm-diameter supply pipes are needed if the average velocity in each must be less than 2.5 m/s?

4.27 You are making beer. The first step is filling the glass carboy with the liquid wort. The internal diameter of the carboy is 15 in., and you wish to fill it up to a depth of 2 ft. If your wort is drawn from the kettle using a siphon process that flows at 3 gpm, how long will it take to fill?

4.28 In your kitchen, the sink is 2 ft by 18 in. by 12 in. deep. You are filling it with water at the rate of 4 gpm. How long will it take (in min) to half fill the sink? After this you turn off the faucet and open the drain slightly so that the tank starts to drain at 1 gpm. What is the rate (in./min) at which the water level drops?

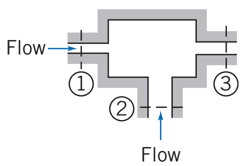
4.29 Ventilation air specifications for classrooms require that at least 8.0 L/s of fresh air be supplied for each person in the room (students and instructor). A system needs to be designed that will supply ventilation air to 6 classrooms, each with a capacity of 20 students. Air enters through a central

duct, with short branches successively leaving for each classroom. Branch registers are 200 mm high and 500 mm wide. Calculate the volume flow rate and air velocity entering each room. Ventilation noise increases with air velocity. Given a supply duct 500 mm high, find the narrowest supply duct that will limit air velocity to a maximum of 1.75 m/s.

4.30 You are trying to pump storm water out of your basement during a storm. The pump can extract 27.5 gpm. The water level in the basement is now sinking by about 4 in./hr. What is the flow rate (gpm) from the storm into the basement? The basement is 30 ft × 20 ft.

4.31 In steady-state flow, downstream the density is 1 kg/m^3, the velocity is 1000 m/sec, and the area is 0.1 m^2. Upstream, the velocity is 1500 m/sec, and the area is 0.25 m^2. What is the density upstream?

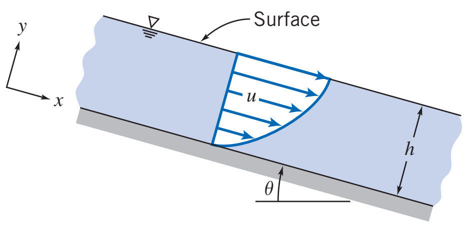
4.32 In the incompressible flow through the device shown, velocities may be considered uniform over the inlet and outlet sections. The following conditions are known: $A_1 = 0.1$ m^2, $A_2 = 0.2$ m^2, $A_3 = 0.15$ m^2, $V_1 = 10e^{-t/2}$ m/s, and $V_2 = 2\cos(2\pi t)$ m/s (t in seconds). Obtain an expression for the velocity at section 3, and plot V_3 as a function of time. At what instant does V_3 first become zero? What is the total mean volumetric flow at section 3?



P4.32

4.33 Oil flows steadily in a thin layer down an inclined plane. The velocity profile is

$$u = \frac{\rho g \sin\theta}{\mu}\left[hy - \frac{y^2}{2}\right]$$



P4.33

Express the mass flow rate per unit width in terms of ρ, μ, g, θ, and h.

4.34 Water enters a wide, flat channel of height $2h$ with a uniform velocity of 2.5 m/s. At the channel outlet the velocity distribution is given by

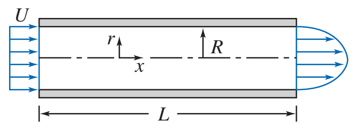
$$\frac{u}{u_{\max}} = 1 - \left(\frac{y}{h}\right)^2$$

where y is measured from the centerline of the channel. Determine the exit centerline velocity, $u_{\max}$.

4.35 Water flows steadily through a pipe of length L and radius $R = 75$ mm. Calculate the uniform inlet velocity, U, if the velocity distribution across the outlet is given by

$$u = u_{\max}\left[1 - \frac{r^2}{R^2}\right]$$

and $u_{\max} = 3$ m/s.



P4.35

4.36 Incompressible fluid flows steadily through a plane diverging channel. At the inlet, of height H, the flow is uniform with magnitude V_1. At the outlet, of height $2H$, the velocity profile is

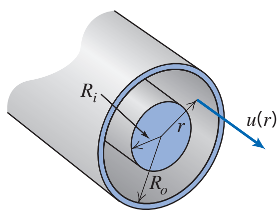
$$V_2 = V_m \cos\left(\frac{\pi y}{2H}\right)$$

where y is measured from the channel centerline. Express V_m in terms of V_1.

4.37 The velocity profile for laminar flow in an annulus is given by

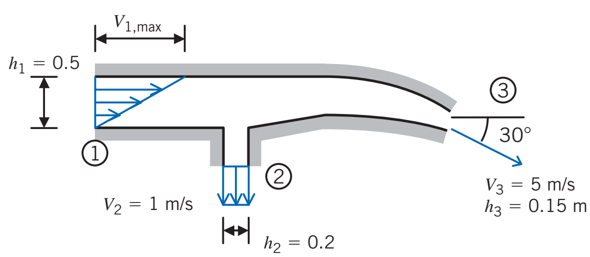
$$u(r) = -\frac{\Delta p}{4\mu L}\left[R_o^2 - r^2 + \frac{R_o^2 - R_i^2}{\ln(R_i/R_o)}\ln\frac{R_o}{r}\right]$$

where $\Delta p/L = -10$ kPa/m is the pressure gradient, μ is the viscosity (SAE 10 oil at 20°C), and $R_o = 5$ mm and $R_i = 1$ mm are the outer and inner radii. Find the volume flow rate, the average velocity, and the maximum velocity. Plot the velocity distribution.



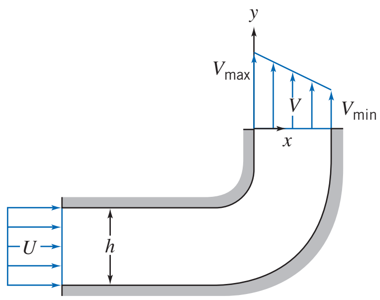
P4.37

4.38 A two-dimensional reducing bend has a linear velocity profile at section 1. The flow is uniform at sections 2 and 3. The fluid is incompressible and the flow is steady. Find the maximum velocity, $V_{1,\max}$, at section 1.



P4.38

4.39 Water enters a two-dimensional, square channel of constant width, $h = 75.5$ mm, with uniform velocity, U. The channel makes a 90° bend that distorts the flow to produce the linear velocity profile shown at the exit, with $v_{max} = 2\ v_{min}$. Evaluate v_{min}, if $U = 7.5$ m/s.



P4.39, 4.80, 4.98

4.40 Viscous liquid from a circular tank, $D = 300$ mm in diameter, drains through a long circular tube of radius $R = 50$ mm. The velocity profile at the tube discharge is

$$u = u_{max}\left[1 - \left(\frac{r}{R}\right)^2\right]$$

Show that the average speed of flow in the drain tube is $\overline{V} = \frac{1}{2}u_{max}$. Evaluate the rate of change of liquid level in the tank at the instant when $u_{max} = 0.155$ m/s.

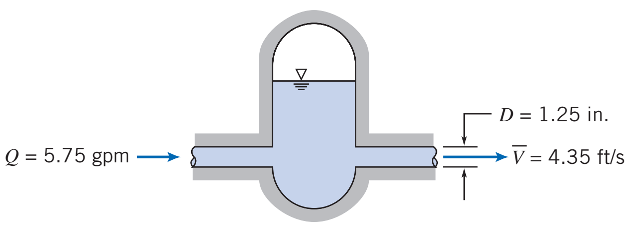
4.41 A porous round tube with $D = 60$ mm carries water. The inlet velocity is uniform with $V_1 = 7.0$ m/s. Water flows radially and axisymmetrically outward through the porous walls with velocity distribution

$$v = V_0\left[1 - \left(\frac{x}{L}\right)^2\right]$$

where $V_0 = 0.03$ m/s and $L = 0.950$ m. Calculate the mass flow rate inside the tube at $x = L$.

4.42 A rectangular tank used to supply water for a Reynolds flow experiment is 230 mm deep. Its width and length are $W = 150$ mm and $L = 230$ mm. Water flows from the outlet tube (inside diameter $D = 6.35$ mm) at Reynolds number $Re = 2000$, when the tank is half full. The supply valve is closed. Find the rate of change of water level in the tank at this instant.

4.43 A *hydraulic accumulator* is designed to reduce pressure pulsations in a machine tool hydraulic system. For the instant shown, determine the rate at which the accumulator gains or loses hydraulic oil.



P4.43

4.44 A cylindrical tank, 0.3 m in diameter, drains through a hole in its bottom. At the instant when the water depth is 0.6 m, the flow rate from the tank is observed to be 4 kg/s. Determine the rate of change of water level at this instant.

4.45 A tank of 0.4 m^3 volume contains compressed air. A valve is opened and air escapes with a velocity of 250 m/s through an opening of 100 mm^2 area. Air temperature passing through the opening is −20°C and the absolute pressure is 300 kPa. Find the rate of change of density of the air in the tank at this moment.

4.46 Air enters a tank through an area of 0.2 ft^2 with a velocity of 15 ft/s and a density of 0.03 slug/ft^3. Air leaves with a velocity of 5 ft/s and a density equal to that in the tank. The initial density of the air in the tank is 0.02 slug/ft^3. The total tank volume is 20 ft^3 and the exit area is 0.4 ft^2. Find the initial rate of change of density in the tank.

4.47 A recent TV news story about lowering Lake Shafer near Monticello, Indiana, by increasing the discharge through the dam that impounds the lake, gave the following information for flow through the dam:

Normal flow rate	290 cfs
Flow rate during draining of lake	2000 cfs

(The flow rate during draining was stated to be equivalent to 16,000 gal/s.) The announcer also said that during draining the lake level was expected to fall at the rate of 1 ft every 8 hr. Calculate the actual flow rate during draining in gal/s. Estimate the surface area of the lake.

4.48 A cylindrical tank, of diameter D = 6 in., drains through an opening, d = 0.25 in., in the bottom of the tank. The speed of the liquid leaving the tank is approximately $V = \sqrt{2gy}$ where y is the height from the tank bottom to the free surface. If the tank is initially filled with water to y_0 = 3 ft, determine the water depths at t = 1 min, t = 2 min, and t = 3 min. Plot y (ft) versus t for the first three min.

4.49 For the conditions of Problem 4.48, estimate the times required to drain the tank from initial depth to a depth $y = 2$ ft (a change in depth of 1 ft), and from $y = 2$ ft to $y = 1$ ft (also a change in depth of 1 ft). Can you explain the discrepancy in these times? Plot the time to drain to a depth y = 1 ft as a function of opening sizes ranging from d = 0.1 in. to 0.5 in.

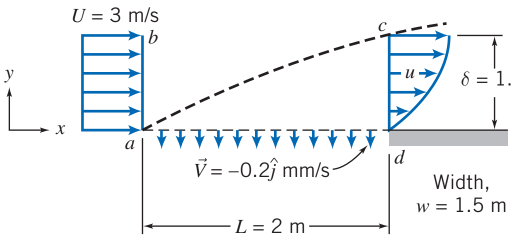
4.50 A conical flask contains water to height $H = 36.8$ mm, where the flask diameter is $D = 29.4$ mm. Water drains out through a smoothly rounded hole of diameter $d = 7.35$ mm at the apex of the cone. The flow speed at the exit is approximately $V = \sqrt{2gy}$, where y is the height of the liquid free surface above the hole. A stream of water flows into the top of the flask at constant volume flow rate, $Q = 3.75 \times 10^{-7}$ m^3/hr. Find the volume flow rate from the bottom of the flask. Evaluate the direction and rate of change of water surface level in the flask at this instant.

4.51 A conical funnel of half-angle $\theta = 15°$, with maximum diameter $D = 70$ mm and height H, drains through a hole (diameter $d = 3.12$ mm) in its bottom. The speed of the liquid leaving the funnel is approximately $V = \sqrt{2gy}$, where y is the height of the liquid free surface above the hole. Find the rate of change of surface level in the funnel at the instant when $y = H/2$.

4.52 Water flows steadily past a porous flat plate. Constant suction is applied along the porous section. The velocity profile at section *cd* is

$$\frac{u}{U_\infty} = 3\left[\frac{y}{\delta}\right] - 2\left[\frac{y}{\delta}\right]^{3/2}$$

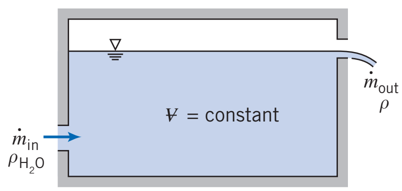
Evaluate the mass flow rate across section *bc*.



P4.52, P4.53

4.53 Consider incompressible steady flow of standard air in a boundary layer on the length of porous surface shown. Assume the boundary layer at the downstream end of the surface has an approximately parabolic velocity profile, $u/U_\infty = 2(y/\delta) - (y/\delta)^2$. Uniform suction is applied along the porous surface, as shown. Calculate the volume flow rate across surface *cd*, through the porous suction surface, and across surface *bc*.

4.54 A tank of fixed volume contains brine with initial density, ρ_i, greater than water. Pure water enters the tank steadily and mixes thoroughly with the brine in the tank. The liquid level in the tank remains constant. Derive expressions for (a) the rate of change of density of the liquid mixture in the tank and (b) the time required for the density to reach the value ρ_f, where $\rho_i > \rho_f > \rho_{H_2O}$.



P4.54

4.55 A conical funnel of half-angle $\theta = 30°$ drains through a small hole of diameter $d = 0.25$ in. at the vertex. The speed of the liquid leaving the funnel is approximately $V = \sqrt{2gy}$, where y is the height of the liquid free surface above the hole. The funnel initially is filled to height $y_0 = 12$ in. Obtain an expression for the time, t, for the funnel to completely drain, and evaluate. Find the time to drain from 12 in. to 6 in. (a change in depth of 6 in.), and from 6 in. to completely empty (also a change in depth of 6 in.). Can you explain the discrepancy in these times? Plot the drain time t as a function diameter d for d ranging from 0.25 in. to 0.5 in.

4.56 For the funnel of Problem 4.55, find the diameter d required if the funnel is to drain in $t = 1$ min. from an initial depth $y_0 = 12$ in. Plot the diameter d required to drain the funnel in 1 min as a function of initial depth y_0, for y_0 ranging from 1 in. to 24 in.

4.57 Over time, air seeps through pores in the rubber of high-pressure bicycle tires. The saying is that a tire loses pressure at the rate of "a pound [1 psi] a day." The true rate of pressure loss is not constant; instead, the instantaneous leakage mass flow rate is proportional to the air density in the tire and to the gage pressure in the tire, $\dot{m} \propto \rho p$. Because the leakage rate is slow, air in the tire is nearly isothermal. Consider a tire that initially is inflated to 0.6 MPa (gage). Assume the initial rate of pressure loss is 1 psi per day. Estimate how long it will take for the pressure to drop to 500 kPa. How accurate is "a pound a day" over the entire 30 day period? Plot the pressure as a function of time over the 30 day period. Show the rule-of-thumb results for comparison.

Momentum Equation for Inertial Control Volume

4.58 Evaluate the net rate of flux of momentum out through the control surface of Problem 4.24.

4.59 For the conditions of Problem 4.34, evaluate the ratio of the x-direction momentum flux at the channel outlet to that at the inlet.

4.60 For the conditions of Problem 4.35, evaluate the ratio of the x-direction momentum flux at the pipe outlet to that at the inlet.

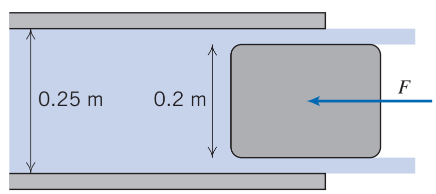
4.61 Evaluate the net momentum flux through the bend of Problem 4.38, if the depth normal to the diagram is $w = 1$ m.

4.62 Evaluate the net momentum flux through the channel of Problem 4.39. Would you expect the outlet pressure to be higher, lower, or the same as the inlet pressure? Why?

4.63 Water jets are being used more and more for metal cutting operations. If a pump generates a flow of 1 gpm through an orifice of 0.01 in. diameter, what is the average jet speed? What force (lbf) will the jet produce at impact, assuming as an approximation that the water sprays sideways after impact?

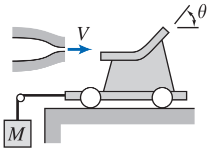
4.64 Considering that in the fully developed region of a pipe, the integral of the axial momentum is the same at all cross sections, explain the reason for the pressure drop along the pipe.

4.65 Find the force required to hold the plug in place at the exit of the water pipe. The flow rate is 1.5 m^3/s, and the upstream pressure is 3.5 MPa.

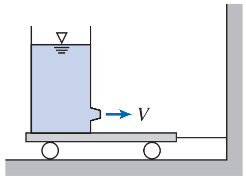


P4.65

4.66 A jet of water issuing from a stationary nozzle at 10 m/s ($A_j = 0.1$ m^2) strikes a turning vane mounted on a cart as shown. The vane turns the jet through angle $\theta = 40°$. Determine the value of M required to hold the cart stationary. If the vane angle θ is adjustable, plot the mass, M, needed to hold the cart stationary versus θ for $0 \le \theta \le 180°$.



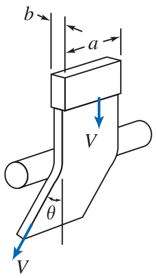
P4.66



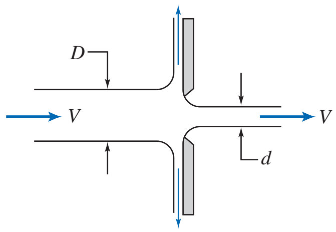
P4.67

4.67 A large tank of height $h = 1$ m and diameter $D = 0.75$ m is affixed to a cart as shown. Water issues from the tank through a nozzle of diameter $d = 15$ mm. The speed of the liquid leaving the tank is approximately $V = \sqrt{2gy}$, where y is the height from the nozzle to the free surface. Determine the tension in the wire when $y = 0.9$ m. Plot the tension in the wire as a function of water depth for $0 \le y \le 0.9$ m.

4.68 A circular cylinder inserted across a stream of flowing water deflects the stream through angle θ, as shown. (This is termed the "Coanda effect.") For $a = 12.5$ mm, $b = 2.5$ mm, $V = 3$ m/s, and $\theta = 20°$, determine the horizontal component of the force on the cylinder caused by the flowing water.



P4.68

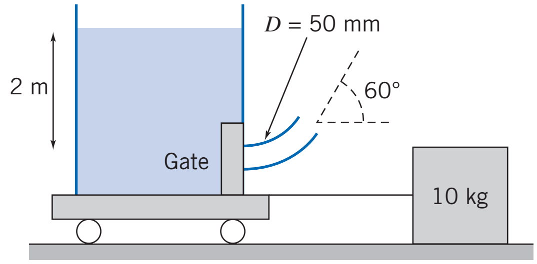


P4.69

4.69 A vertical plate has a sharp-edged orifice at its center. A water jet of speed V strikes the plate concentrically. Obtain an expression for the external force needed to hold the plate in place, if the jet leaving the orifice also has speed V. Evaluate the force for $V = 15$ ft/s, $D = 4$ in., and $d = 1$ in. Plot the required force as a function of diameter ratio for a suitable range of diameter d.

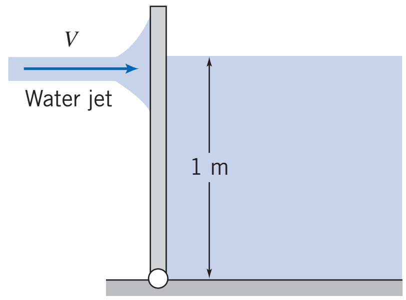
4.70 In a laboratory experiment, the water flow rate is to be measured catching the water as it vertically exits a pipe into an empty open tank that is on a zeroed balance. The tank is 10 m directly below the pipe exit, and the pipe diameter is 50 mm. One student obtains a flow rate by noting that after 60 s the volume of water (at 4°C) in the tank was 2 m^3. Another student obtains a flow rate by reading the instantaneous weight accumulated of 3150 kg indicated at the 60-s point. Find the mass flow rate each student computes. Why do they disagree? Which one is more accurate? Show that the magnitude of the discrepancy can be explained by any concept you may have.

4.71 A tank of water sits on a cart with frictionless wheels as shown. The cart is attached using a cable to a mass $M = 10$ kg, and the coefficient of static friction of the mass with the ground is $\mu = 0.55$. If the gate blocking the tank exit is removed, will the resulting exit flow be sufficient to start the tank moving? (Assume the water flow is frictionless, and that the jet velocity is $V = \sqrt{2gh}$, where $h = 2$ m is the water depth.) Find the mass M that is just sufficient to hold the tank in place.



P4.71

4.72 A gate is 1 m wide and 1.2 m tall and hinged at the bottom. On one side the gate holds back a 1-m-deep body of water. On the other side, a 5-cm diameter water jet hits the gate at a height of 1 m. What jet speed V is required to hold the gate vertical? What will the required speed be if the body of water is lowered to 0.5 m? What will the required speed be if the water level is lowered to 0.25 m?

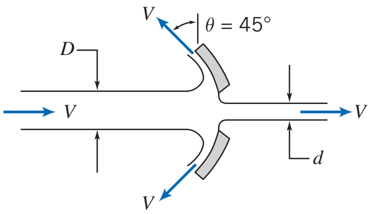


P4.72

4.73 A farmer purchases 675 kg of bulk grain from the local co-op. The grain is loaded into his pickup truck from a hopper with an outlet diameter of 0.3 m. The loading operator determines the payload by observing the indicated gross mass of the truck as a function of time. The grain flow from the hopper ($\dot{m} = 40\,\text{kg/s}$) is terminated when the indicated scale reading reaches the desired gross mass. If the grain density is 600 kg/m^3, determine the true payload.

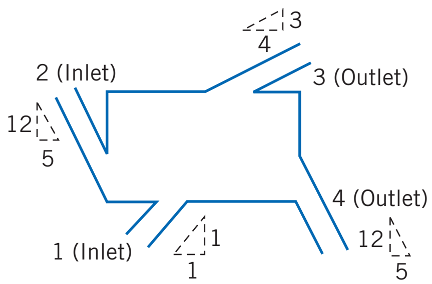
4.74 Water flows steadily through a fire hose and nozzle. The hose is 75 mm inside diameter, and the nozzle tip is 25 mm ID; water gage pressure in the hose is 510 kPa, and the stream leaving the nozzle is uniform. The exit speed and pressure are 32 m/s and atmospheric, respectively. Find the force transmitted by the coupling between the nozzle and hose. Indicate whether the coupling is in tension or compression.

4.75 A shallow circular dish has a sharp-edged orifice at its center. A water jet, of speed V, strikes the dish concentrically. Obtain an expression for the external force needed to hold the dish in place if the jet issuing from the orifice also has speed V. Evaluate the force for $V = 5$ m/s, $D = 100$ mm, and $d = 25$ mm. Plot the required force as a function of the angle θ ($0 \le \theta \le 90°$) with diameter ratio as a parameter for a suitable range of diameter d.



P4.75

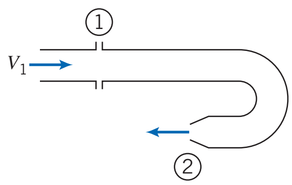
4.76 Obtain expressions for the rate of change in mass of the control volume shown, as well as the horizontal and vertical forces required to hold it in place, in terms of p_1, A_1, V_1, p_2, A_2, V_2, p_3, A_3, V_3, p_4, A_4, V_4, and the constant density ρ.



P4.76

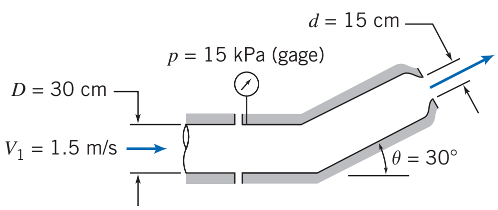
4.77 A 180° elbow takes in water at an average velocity of 0.8 m/s and a pressure of 350 kPa (gage) at the inlet, where the diameter is 0.2 m. The exit pressure is 75 kPa, and the diameter is 0.04 m. What is the force required to hold the elbow in place?

4.78 Water is flowing steadily through the 180° elbow shown. At the inlet to the elbow the gage pressure is 15 psi. The water discharges to atmospheric pressure. Assume properties are uniform over the inlet and outlet areas: $A_1 = 4$ in.^{2}, $A_2 = 1$ in.^{2}, and $V_1 = 10$ ft/s. Find the horizontal component of force required to hold the elbow in place.



P4.78

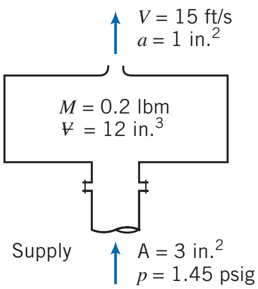
4.79 Water flows steadily through the nozzle shown, discharging to atmosphere. Calculate the horizontal component of force in the flanged joint. Indicate whether the joint is in tension or compression.



P4.79

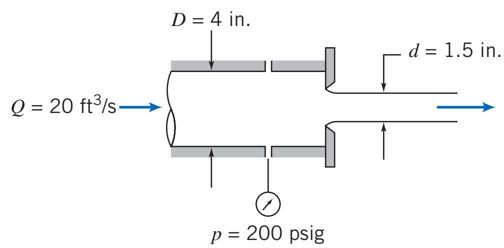
4.80 Assume the bend of Problem 4.39 is a segment of a larger channel and lies in a horizontal plane. The inlet pressure is 170 kPa (abs), and the outlet pressure is 130 kPa (abs). Find the force required to hold the bend in place.

4.81 A spray system is shown in the diagram. Water is supplied at $p = 1.45$ psig, through the flanged opening of area $A = 3$ in.^{2} The water leaves in a steady free jet at atmospheric pressure. The jet area and speed are $a = 1.0$ in.^{2} and $V = 15$ ft/s. The mass of the spray system is 0.2 lbm and it contains $\forall = 12$ in.^{3} of water. Find the force exerted on the supply pipe by the spray system.



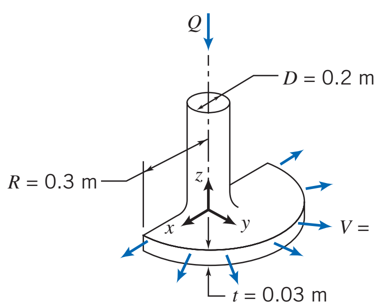
P4.81

4.82 A flat plate orifice of 2 in. diameter is located at the end of a 4-in.-diameter pipe. Water flows through the pipe and orifice at 20 ft^3/s. The diameter of the water jet downstream from the orifice is 1.5 in. Calculate the external force required to hold the orifice in place. Neglect friction on the pipe wall.



P4.82

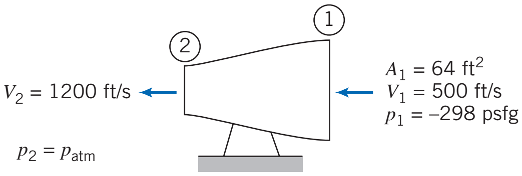
4.83 The nozzle shown discharges a sheet of water through a 180° arc. The water speed is 15 m/s and the jet thickness is 30 mm at a radial distance of 0.3 m from the centerline of the supply pipe. Find (a) the volume flow rate of water in the jet sheet and (b) the y component of force required to hold the nozzle in place.



P4.83

4.84 At rated thrust, a liquid-fueled rocket motor consumes 80 kg/s of nitric acid as oxidizer and 32 kg/s of aniline as fuel. Flow leaves axially at 180 m/s relative to the nozzle and at 110 kPa. The nozzle exit diameter is $D = 0.6$ m. Calculate the thrust produced by the motor on a test stand at standard sea-level pressure.

4.85 A typical jet engine test stand installation is shown, together with some test data. Fuel enters the top of the engine vertically at a rate equal to 2 percent of the mass flow rate of the inlet air. For the given conditions, compute the air flow rate through the engine and estimate the thrust.

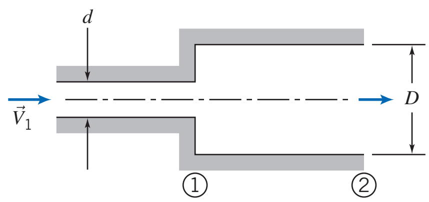


P4.85

4.86 Consider flow through the sudden expansion shown. If the flow is incompressible and friction is neglected, show that the pressure rise, $\Delta p = p_2 - p_1$, is given by

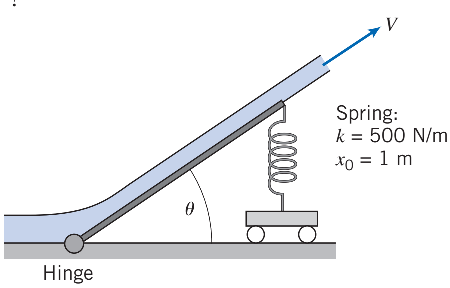
$$\frac{\Delta p}{\frac{1}{2}\rho \bar{V}_1^2} = 2\left(\frac{d}{D}\right)^2\left[1 - \left(\frac{d}{D}\right)^2\right]$$

Plot the nondimensional pressure rise versus diameter ratio to determine the optimum value of d/D and the corresponding value of the nondimensional pressure rise. *Hint:* Assume the pressure is uniform and equal to p_1 on the vertical surface of the expansion.



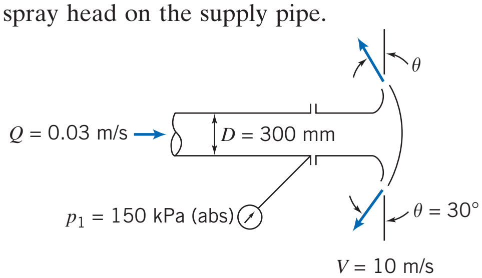
P4.86

4.87 A free jet of water with constant cross-section area 0.01 m^2 is deflected by a hinged plate of length 2 m supported by a spring with spring constant $k = 500$ N/m and uncompressed length $x_0 = 1$ m. Find and plot the deflection angle θ as a function of jet speed V. What jet speed has a deflection of $\theta = 5°$?



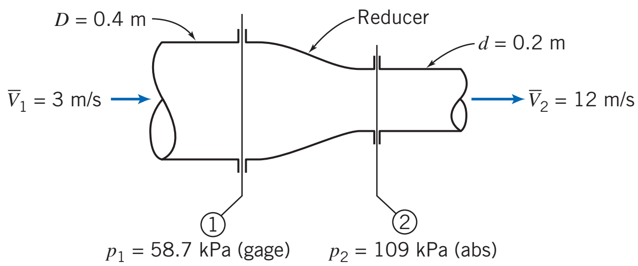
P4.87

4.88 A conical spray head is shown. The fluid is water and the exit stream is uniform. Evaluate (a) the thickness of the spray sheet at 400 mm radius and (b) the axial force exerted by the spray head on the supply pipe.



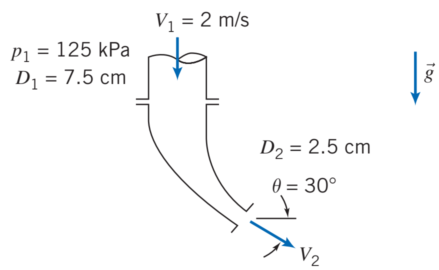
P4.88

4.89 A reducer in a piping system is shown. The internal volume of the reducer is 0.2 m^3 and its mass is 25 kg. Evaluate the total force that must be provided by the surrounding pipes to support the reducer. The fluid is gasoline.



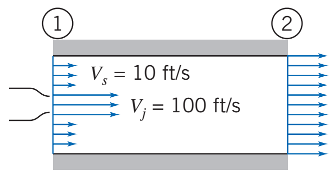
P4.89

4.90 A curved nozzle assembly that discharges to the atmosphere is shown. The nozzle mass is 4.5 kg and its internal volume is 0.002 m^3. The fluid is water. Determine the reaction force exerted by the nozzle on the coupling to the inlet pipe.



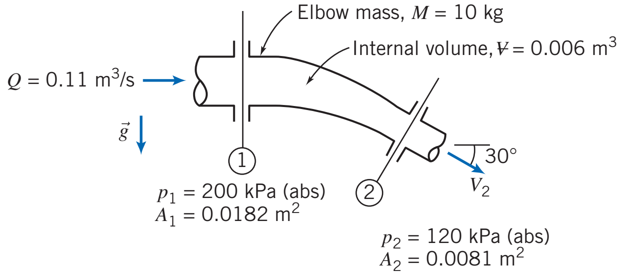
P4.90

4.91 A water jet pump has jet area 0.1 ft^2 and jet speed 100 ft/s. The jet is within a secondary stream of water having speed $V_s = 10$ ft/s. The total area of the duct (the sum of the jet and secondary stream areas) is 0.75 ft^2. The water is thoroughly mixed and leaves the jet pump in a uniform stream. The pressures of the jet and secondary stream are the same at the pump inlet. Determine the speed at the pump exit and the pressure rise, $p_2 - p_1$.



P4.91

4.92 A 30° reducing elbow is shown. The fluid is water. Evaluate the components of force that must be provided by the adjacent pipes to keep the elbow from moving.



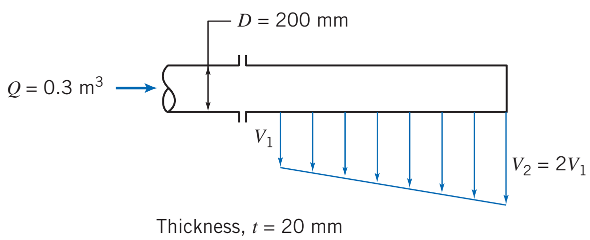
P4.92

4.93 Consider the steady adiabatic flow of air through a long straight pipe with 0.05 m^2 cross-sectional area. At the inlet, the air is at 200 kPa (gage), 60°C, and has a velocity of 150 m/s. At the exit, the air is at 80 kPa and has a velocity of 300 m/s. Calculate the axial force of the air on the pipe. (Be sure to make the direction clear.)

4.94 A monotube boiler consists of a 20 ft length of tubing with 0.375 in. inside diameter. Water enters at the rate of 0.3 lbm/s at 500 psia. Steam leaves at 400 psig with 0.024 slug/ft^3 density. Find the magnitude and direction of the force exerted by the flowing fluid on the tube.

4.95 A gas flows steadily through a heated porous pipe of constant 0.15 m^2 cross-sectional area. At the pipe inlet, the absolute pressure is 400 kPa, the density is 6 kg/m^3, and the mean velocity is 170 m/s. The fluid passing through the porous wall leaves in a direction normal to the pipe axis, and the total flow rate through the porous wall is 20 kg/s. At the pipe outlet, the absolute pressure is 300 kPa and the density is 2.75 kg/m^3. Determine the axial force of the fluid on the pipe.

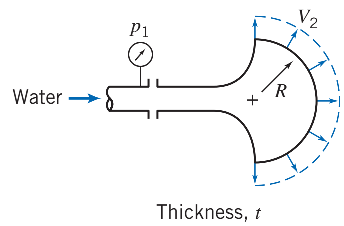
4.96 Water is discharged at a flow rate of 0.3 m^3/s from a narrow slot in a 200-mm-diameter pipe. The resulting horizontal two-dimensional jet is 1 m long and 20 mm thick, but of nonuniform velocity; the velocity at location 2 is twice that at location 1. The pressure at the inlet section is 50 kPa (gage). Calculate (a) the velocity in the pipe and at locations 1 and 2 and (b) the forces required at the coupling to hold the spray pipe in place. Neglect the mass of the pipe and the water it contains.



P4.96

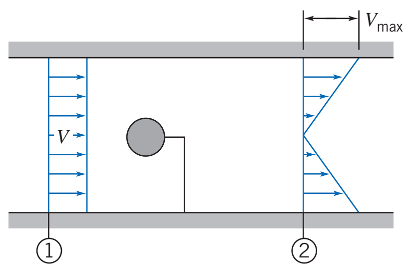
4.97 Water flows steadily through the square bend of Problem 4.39. Flow at the inlet is at p_1 = 185 kPa (abs). Flow at the exit is nonuniform, vertical, and at atmospheric pressure. The mass of the channel structure is M_c = 2.05 kg; the internal volume of the channel is $\forall$ = 0.00355 m^3. Evaluate the force exerted by the channel assembly on the supply duct.

4.98 A nozzle for a spray system is designed to produce a flat radial sheet of water. The sheet leaves the nozzle at V_2 = 10 m/s, covers 180° of arc, and has thickness t = 1.5 mm. The nozzle discharge radius is R = 50 mm. The water supply pipe is 35 mm in diameter and the inlet pressure is p_1 = 150 kPa (abs). Evaluate the axial force exerted by the spray nozzle on the coupling.



P4.98

4.99 A small round object is tested in a 0.75-m diameter wind tunnel. The pressure is uniform across sections 1 and 2. The upstream pressure is 30 mm H_2O (gage), the downstream pressure is 15 mm H_2O (gage), and the mean air speed is 12.5 m/s. The velocity profile at section 2 is linear; it varies from zero at the tunnel centerline to a maximum at the tunnel wall. Calculate (a) the mass flow rate in the wind tunnel, (b) the maximum velocity at section 2, and (c) the drag of the object and its supporting vane. Neglect viscous resistance at the tunnel wall.



P4.99

4.100 The horizontal velocity in the wake behind an object in an air stream of velocity U is given by

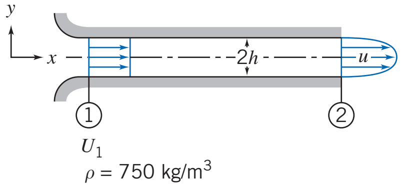
$$u(r) = U\left[1 - \cos^2\left(\frac{\pi r}{2}\right)\right] \qquad |r| \le 1$$
$$u(r) = U \qquad |r| > 1$$

where r is the nondimensional radial coordinate, measured perpendicular to the flow. Find an expression for the drag on the object.

4.101 An incompressible fluid flows steadily in the entrance region of a two-dimensional channel of height $2h$ = 100 mm and width w = 25 mm. The flow rate is Q = 0.025 m^3/s. Find the uniform velocity U_1 at the entrance. The velocity distribution at a section downstream is

$$\frac{u}{u_{max}} = 1 - \left(\frac{y}{h}\right)^2$$

Evaluate the maximum velocity at the downstream section. Calculate the pressure drop that would exist in the channel if viscous friction at the walls could be neglected.

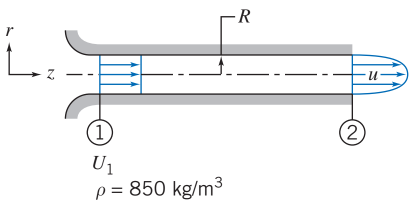


P4.101

4.102 An incompressible fluid flows steadily in the entrance region of a circular tube of radius $R = 75$ mm. The flow rate is $Q = 0.1\ \text{m}^3/\text{s}$. Find the uniform velocity U_1 at the entrance. The velocity distribution at a section downstream is

$$\frac{u}{u_{\max}} = 1 - \left(\frac{r}{R}\right)^2$$

Evaluate the maximum velocity at the downstream section. Calculate the pressure drop that would exist in the channel if viscous friction at the walls could be neglected.

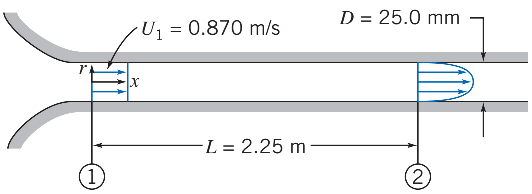


P4.102

4.103 Air enters a duct, of diameter $D = 25.0$ mm, through a well-rounded inlet with uniform speed, $U_1 = 0.870$ m/s. At a downstream section where $L = 2.25$ m, the fully developed velocity profile is

$$\frac{u(r)}{U_c} = 1 - \left(\frac{r}{R}\right)^2$$

The pressure drop between these sections is $p_1 - p_2 = 1.92\ \text{N/m}^2$. Find the total force of friction exerted by the tube on the air.



P4.103

4.104 Consider the incompressible flow of fluid in a boundary layer as depicted in Example 4.2. Show that the friction drag force of the fluid on the surface is given by

$$F_f = \int_0^{\delta} \rho u(U - u) w\, dy$$

Evaluate the drag force for the conditions of Example 4.2.

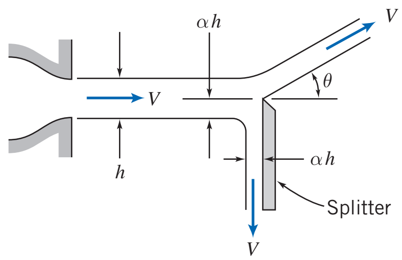
4.105 A fluid with density $\rho = 750\ \text{kg/m}^3$ flows along a flat plate of width 1 m. The undisturbed freestream speed is $U_0 = 10$ m/s. At $L = 1$ m downstream from the leading edge of the plate, the boundary-layer thickness is $\delta = 5$ mm. The velocity profile at this location is

$$\frac{u}{U_0} = \frac{3}{2}\frac{y}{\delta} - \frac{1}{2}\left(\frac{y}{\delta}\right)^3$$

Plot the velocity profile. Calculate the horizontal component of force required to hold the plate stationary.

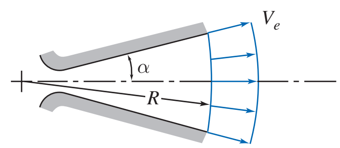
4.106 Air at standard conditions flows along a flat plate. The undisturbed freestream speed is $U_0 = 20$ m/s. At $L = 0.4$ m downstream from the leading edge of the plate, the boundary-layer thickness is $\delta = 2$ mm. The velocity profile at this location is approximated as $u/U_0 = y/\delta$. Calculate the horizontal component of force per unit width required to hold the plate stationary.

4.107 A sharp-edged splitter plate inserted part way into a flat stream of flowing water produces the flow pattern shown. Analyze the situation to evaluate θ as a function of α, where $0 \leq \alpha < 0.5$. Evaluate the force needed to hold the splitter plate in place. (Neglect any friction force between the water stream and the splitter plate.) Plot both θ and R_x as functions of α.



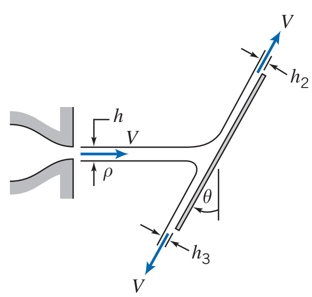
P4.107

4.108 Gases leaving the propulsion nozzle of a rocket are modeled as flowing radially outward from a point upstream from the nozzle throat. Assume the speed of the exit flow, V_e, has constant magnitude. Develop an expression for the axial thrust, T_a, developed by flow leaving the nozzle exit plane. Compare your result to the one-dimensional approximation, $T = \dot{m}V_e$. Evaluate the percent error for $\alpha = 15°$. Plot the percent error versus α for $0 \leq \alpha \leq 22.5°$.



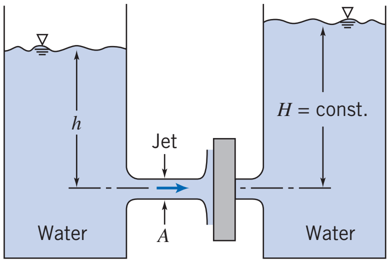
P4.108

4.109 When a plane liquid jet strikes an inclined flat plate, it splits into two streams of equal speed but unequal thickness. For frictionless flow there can be no tangential force on the plate surface. Use this assumption to develop an expression for h_2/h as a function of plate angle, θ. Plot your results and comment on the limiting cases, $\theta = 0$ and $\theta = 90°$.



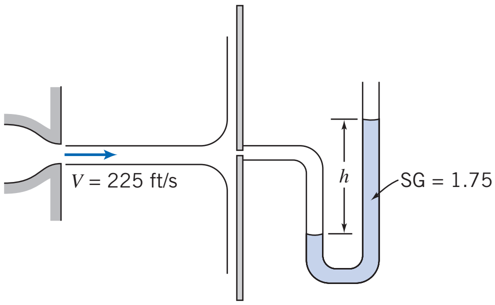
P4.109

***4.110** Two large tanks containing water have small smoothly contoured orifices of equal area. A jet of liquid issues from the left tank. Assume the flow is uniform and unaffected by friction. The jet impinges on a vertical flat plate covering the opening of the right tank. Determine the minimum value for the height, h, required to keep the plate in place over the opening of the right tank.



P4.110

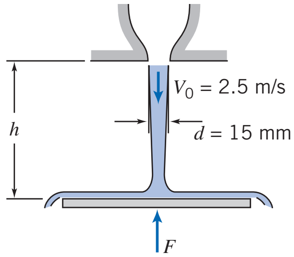
***4.111** A horizontal axisymmetric jet of air with 0.5 in. diameter strikes a stationary vertical disk of 8 in. diameter. The jet speed is 225 ft/s at the nozzle exit. A manometer is connected to the center of the disk. Calculate (a) the deflection, h, if the manometer liquid has SG = 1.75 and (b) the force exerted by the jet on the disk.



P4.111

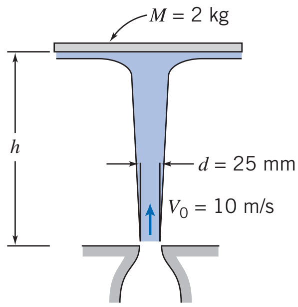
***4.112** Students are playing around with a water hose. When they point it straight up, the water jet just reaches one of the windows of Professor Pritchard's office, 10 m above. If the hose diameter is 1 cm, estimate the water flow rate (L/min). Professor Pritchard happens to come along and places his hand just above the hose to make the jet spray sideways axisymmetrically. Estimate the maximum pressure, and the total force, he feels. The next day the students again are playing around, and this time aim at Professor Fox's window, 15 m above. Find the flow rate (L/min) and the total force and maximum pressure when he, of course, shows up and blocks the flow.

***4.113** A uniform jet of water leaves a 15-mm-diameter nozzle and flows directly downward. The jet speed at the nozzle exit plane is 2.5 m/s. The jet impinges on a horizontal disk and flows radially outward in a flat sheet. Obtain a general expression for the velocity the liquid stream would reach at the level of the disk. Develop an expression for the force required to hold the disk stationary, neglecting the mass of the disk and water sheet. Evaluate for h = 3 m.



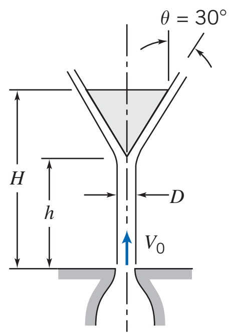
P4.113

***4.114** A 2-kg disk is constrained horizontally but is free to move vertically. The disk is struck from below by a vertical jet of water. The speed and diameter of the water jet are 10 m/s and 25 mm at the nozzle exit. Obtain a general expression for the speed of the water jet as a function of height, h. Find the height to which the disk will rise and remain stationary.



P4.114

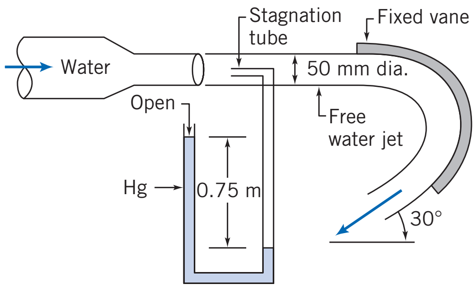
***4.115** Water from a jet of diameter D is used to support the cone-shaped object shown. Derive an expression for the combined mass of the cone and water, M, that can be supported by the jet, in terms of parameters associated with a suitably chosen control volume. Use your expression to calculate M when $V_0 = 10$ m/s, $H = 1$ m, $h = 0.8$ m, $D = 50$ mm, and $\theta = 30°$. Estimate the mass of water in the control volume.



P4.115

***4.116** A stream of water from a 50-mm-diameter nozzle strikes a curved vane, as shown. A stagnation tube connected to a mercury-filled U-tube manometer is located in the nozzle exit plane. Calculate the speed of the water leaving the nozzle. Estimate the horizontal component of force exerted on the vane by the jet. Comment on each assumption used to solve this problem.

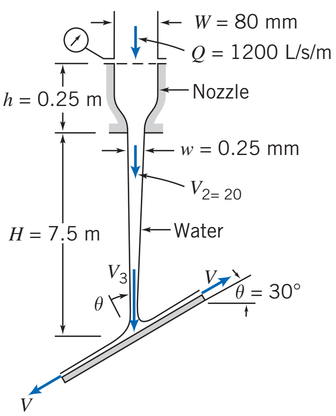
*These problems require material from sections that may be omitted without loss of continuity in the text material.



P4.116

***4.117** A Venturi meter installed along a water pipe consists of a convergent section, a constant-area throat, and a divergent section. The pipe diameter is $D = 100$ mm, and the throat diameter is $d = 50$ mm. Find the net fluid force acting on the convergent section if the water pressure in the pipe is 200 kPa (gage) and the flow rate is 1000 L/min. For this analysis, neglect viscous effects.

***4.118** A plane nozzle discharges vertically 1200 L/s per unit width downward to atmosphere. The nozzle is supplied with a steady flow of water. A stationary, inclined, flat plate, located beneath the nozzle, is struck by the water stream. The water stream divides and flows along the inclined plate; the two streams leaving the plate are of unequal thickness. Frictional effects are negligible in the nozzle and in the flow along the plate surface. Evaluate the minimum gage pressure required at the nozzle inlet.



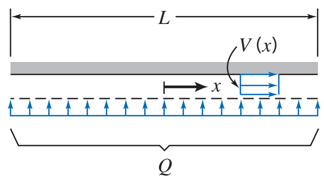
P4.118

***4.119** You turn on the kitchen faucet very slightly, so that a very narrow stream of water flows into the sink. You notice that it is "glassy" (laminar flow) and gets narrower and remains "glassy" for about the first 50 mm of descent. When you measure the flow, it takes three min to fill a 1-L bottle, and you estimate the stream of water is initially 5 mm in diameter. Assuming the speed at any cross section is uniform and neglecting viscous effects, derive expressions for and plot the variations of stream speed and diameter as functions of z (take the origin of coordinates at the faucet exit). What are the speed and diameter when it falls to the 50-mm point?

***4.120** In ancient Egypt, circular vessels filled with water sometimes were used as crude clocks. The vessels were shaped in such a way that, as water drained from the bottom, the surface level dropped at constant rate, s. Assume that water drains from a small hole of area A. Find an expression for the radius of the vessel, r, as a function of the water level, h. Obtain an expression for the volume of water needed so that the clock will operate for n hours.

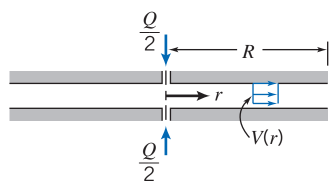
***4.121** A stream of incompressible liquid moving at low speed leaves a nozzle pointed directly downward. Assume the speed at any cross section is uniform and neglect viscous effects. The speed and area of the jet at the nozzle exit are V_0 and A_0, respectively. Apply conservation of mass and the momentum equation to a differential control volume of length dz in the flow direction. Derive expressions for the variations of jet speed and area as functions of z. Evaluate the distance at which the jet area is half its original value. (Take the origin of coordinates at the nozzle exit.)

***4.122** Incompressible fluid of negligible viscosity is pumped, at total volume flow rate Q, through a porous surface into the small gap between closely spaced parallel plates as shown. The fluid has only horizontal motion in the gap. Assume uniform flow across any vertical section. Obtain an expression for the pressure variation as a function of x. *Hint:* Apply conservation of mass and the momentum equation to a differential control volume of thickness dx, located at position x.



P4.122

***4.123** Incompressible liquid of negligible viscosity is pumped, at total volume flow rate Q, through two small holes into the narrow gap between closely spaced parallel disks as shown. The liquid flowing away from the holes has only radial motion. Assume uniform flow across any vertical section and discharge to atmospheric pressure at $r = R$. Obtain an expression for the pressure variation and plot as a function of radius. *Hint:* Apply conservation of mass and the momentum equation to a differential control volume of thickness dr located at radius r.



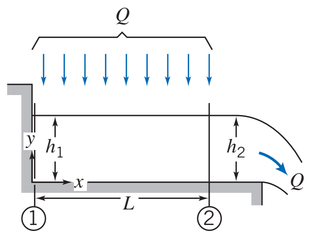
P4.123

***4.124** The narrow gap between two closely spaced circular plates initially is filled with incompressible liquid. At $t = 0$ the upper plate, initially h_0 above the lower plate, begins to move downward toward the lower plate with constant speed, V_0, causing the liquid to be squeezed from the narrow gap. Neglecting viscous effects and assuming uniform flow in the radial direction, develop an expression for the velocity field between the parallel plates. *Hint*: Apply conservation of mass to a control volume with the outer surface located at radius r. Note that even though the speed of the upper plate is

*These problems require material from sections that may be omitted without loss of continuity in the text material.

constant, the flow is unsteady. For $V_0 = 0.01$ m/s and $h_0 = 2$ mm, find the velocity at the exit radius $R = 100$ mm at $t = 0$ and $t = 0.1$ s. Plot the exit velocity as a function of time, and explain the trend.

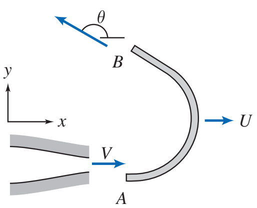
***4.125** Liquid falls vertically into a short horizontal rectangular open channel of width b. The total volume flow rate, Q, is distributed uniformly over area bL. Neglect viscous effects. Obtain an expression for h_1 in terms of h_2, Q, and b. *Hint:* Choose a control volume with outer boundary located at $x = L$. Sketch the surface profile, $h(x)$. *Hint:* Use a differential control volume of width dx.



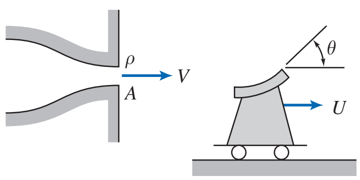
P4.125

***4.126** Design a clepsydra (Egyptian water clock)—a vessel from which water drains by gravity through a hole in the bottom and which indicates time by the level of the remaining water. Specify the dimensions of the vessel and the size of the drain hole; indicate the amount of water needed to fill the vessel and the interval at which it must be filled. Plot the vessel radius as a function of elevation.

4.127 A jet of water is directed against a vane, which could be a blade in a turbine or in any other piece of hydraulic machinery. The water leaves the stationary 40-mm-diameter nozzle with a speed of 25 m/s and enters the vane tangent to the surface at A. The inside surface of the vane at B makes angle $\theta = 150°$ with the x direction. Compute the force that must be applied to maintain the vane speed constant at $U = 5$ m/s.



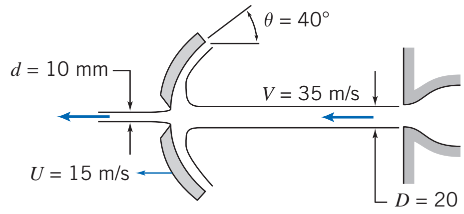
P4.127



P4.128, P4.131, P4.133, P4.145

4.128 Water from a stationary nozzle impinges on a moving vane with turning angle $\theta = 120°$. The vane moves away from the nozzle with constant speed, $U = 10$ m/s, and receives a jet that leaves the nozzle with speed $V = 30$ m/s. The nozzle has an exit area of 0.004 m^2. Find the force that must be applied to maintain the vane speed constant.

4.129 The circular dish, whose cross section is shown, has an outside diameter of 0.20 m. A water jet with speed of 35 m/s strikes the dish concentrically. The dish moves to the left at 15 m/s. The jet diameter is 20 mm. The dish has a hole at its center that allows a stream of water 10 mm in diameter to pass through without resistance. The remainder of the jet is deflected and flows along the dish. Calculate the force required to maintain the dish motion.



P4.129

4.130 A freshwater jet boat takes in water through side vents and ejects it through a nozzle of diameter $D = 75$ mm; the jet speed is V_j. The drag on the boat is given by $F_{\text{drag}} \propto kV^2$, where V is the boat speed. Find an expression for the steady speed, V, in terms of water density ρ, flow rate through the system of Q, constant k, and jet speed V_j. A jet speed $V_j = 15$ m/s produces a boat speed of $V = 10$ m/s.

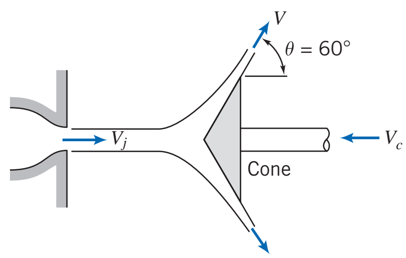
(a) Under these conditions, what is the new flow rate Q?
(b) Find the value of the constant k.
(c) What speed V will be produced if the jet speed is increased to $V_j = 25$ m/s?
(d) What will be the new flow rate?

4.131 A jet of oil (SG = 0.8) strikes a curved blade that turns the fluid through angle $\theta = 180°$. The jet area is 1200 mm^2 and its speed relative to the stationary nozzle is 20 m/s. The blade moves toward the nozzle at 10 m/s. Determine the force that must be applied to maintain the blade speed constant.

4.132 The Canadair CL-215T amphibious aircraft is specially designed to fight fires. It is the only production aircraft that can scoop water—1620 gallons in 12 seconds—from any lake, river, or ocean. Determine the added thrust required during water scooping, as a function of aircraft speed, for a reasonable range of speeds.

4.133 Consider a single vane, with turning angle θ, moving horizontally at constant speed, U, under the influence of an impinging jet as in Problem 4.128. The absolute speed of the jet is V. Obtain general expressions for the resultant force and power that the vane could produce. Show that the power is maximized when $U = V/3$.

4.134 Water, in a 4-in. diameter jet with speed of 100 ft/s to the right, is deflected by a cone that moves to the left at 45 ft/s. Determine (a) the thickness of the jet sheet at a radius of 9 in. and (b) the external horizontal force needed to move the cone.

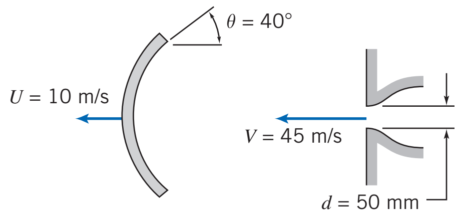


P4.134

4.135 The circular dish, whose cross section is shown, has an outside diameter of 0.15 m. A water jet strikes the dish concentrically and then flows outward along the surface of the dish. The jet speed is 45 m/s and the dish moves to the left at 10 m/s. Find the thickness of the jet sheet at a radius of

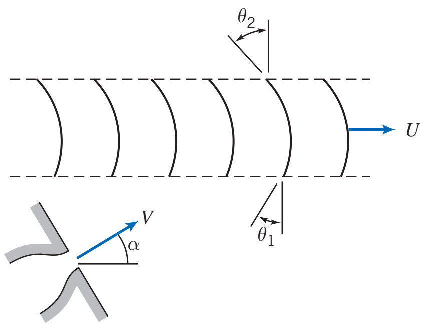
*These problems require material from sections that may be omitted without loss of continuity in the text material.

75 mm from the jet axis. What horizontal force on the dish is required to maintain this motion?



P4.135

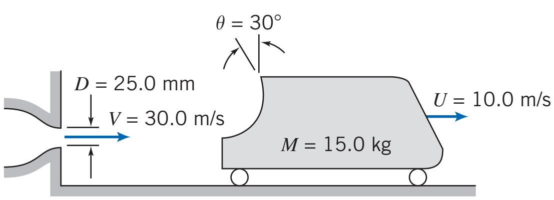
4.136 Consider a series of turning vanes struck by a continuous jet of water that leaves a 50-mm diameter nozzle at constant speed, $V = 86.6$ m/s. The vanes move with constant speed, $U = 50$ m/s. Note that all the mass flow leaving the jet crosses the vanes. The curvature of the vanes is described by angles $\theta_1 = 30°$ and $\theta_2 = 45°$, as shown. Evaluate the nozzle angle, α, required to ensure that the jet enters tangent to the leading edge of each vane. Calculate the force that must be applied to maintain the vane speed constant.



P4.136, P4.137

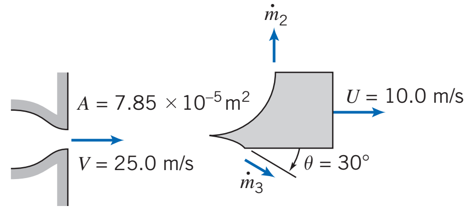
4.137 Consider again the moving multiple-vane system described in Problem 4.136. Assuming that a way could be found to make α nearly zero (and thus, θ_1 nearly 90°), evaluate the vane speed, U, that would result in maximum power output from the moving vane system.

4.138 A steady jet of water is used to propel a small cart along a horizontal track as shown. Total resistance to motion of the cart assembly is given by $F_D = kU^2$, where $k = 0.92\ \mathrm{N \cdot s^2/m^2}$. Evaluate the acceleration of the cart at the instant when its speed is $U = 10$ m/s.



P4.138, P4.140, P4.144

4.139 A plane jet of water strikes a splitter vane and divides into two flat streams, as shown. Find the mass flow rate ratio, $\dot{m}_2/\dot{m}_3$, required to produce zero net vertical force on the splitter vane. If there is a resistive force of 16 N applied to the splitter vane, find the steady speed U of the vane.

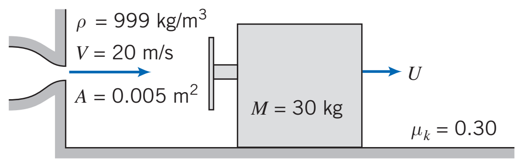


P4.139

Momentum Equation for Control Volume with Rectilinear Acceleration

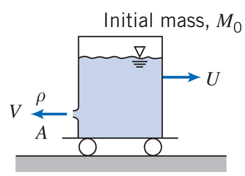
4.140 The hydraulic catapult of Problem 4.138 is accelerated by a jet of water that strikes the curved vane. The cart moves along a level track with negligible resistance. At any time its speed is U. Calculate the time required to accelerate the cart from rest to $U = V/2$.

4.141 A vane/slider assembly moves under the influence of a liquid jet as shown. The coefficient of kinetic friction for motion of the slider along the surface is $\mu_k = 0.30$. Calculate the terminal speed of the slider.



P4.141, P4.143, P4.152, P4.153

4.142 A cart is propelled by a liquid jet issuing horizontally from a tank as shown. The track is horizontal; resistance to motion may be neglected. The tank is pressurized so that the jet speed may be considered constant. Obtain a general expression for the speed of the cart as it accelerates from rest. If $M_0 = 100$ kg, $\rho = 999\ \mathrm{kg/m^3}$, and $A = 0.005\ \mathrm{m^2}$, find the jet speed V required for the cart to reach a speed of 1.5 m/s after 30 seconds. For this condition, plot the cart speed U as a function of time. Plot the cart speed after 30 seconds as a function of jet speed.



P4.142, P4.184

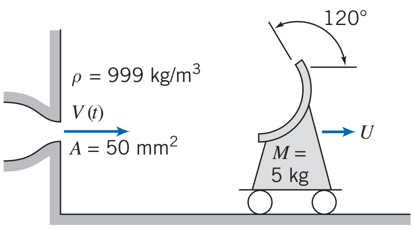
4.143 For the vane/slider problem of Problem 4.141, find and plot expressions for the acceleration and speed of the slider as a function of time.

4.144 If the cart of Problem 4.138 is released at $t = 0$, when would you expect the acceleration to be maximum? Sketch what you would expect for the curve of acceleration versus time. What value of θ would maximize the acceleration at

any time? Why? Will the cart speed ever equal the jet speed? Explain briefly.

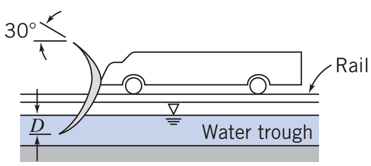
4.145 The acceleration of the vane/cart assembly of Problem 4.128 is to be controlled as it accelerates from rest by changing the vane angle, θ. A constant acceleration, $a = 1.5\ \text{m/s}^2$, is desired. The water jet leaves the nozzle of area $A = 0.025\ \text{m}^2$, with speed $V = 15$ m/s. The vane/cart assembly has a mass of 55 kg; neglect friction. Determine θ at $t = 5$ s. Plot $\theta(t)$ for the given constant acceleration over a suitable range of t.

4.146 The wheeled cart shown rolls with negligible resistance. The cart is to accelerate to the right at a constant rate of $2.5\ \text{m/s}^2$. This is to be accomplished by "programming" the water jet speed, $V(t)$, that hits the cart. The jet area remains constant at $50\ \text{mm}^2$. Find the initial jet speed, and the jet speed and cart speeds after 2.5 s and 5 s. Theoretically, what happens to the value of $(V - U)$ over time?



P4.146

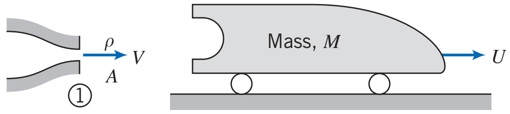
4.147 A rocket sled, weighing 10,000 lbf and traveling 600 mph, is to be braked by lowering a scoop into a water trough. The scoop is 6 in. wide. Determine the time required (after lowering the scoop to a depth of 3 in. into the water) to bring the sled to a speed of 20 mph. Plot the sled speed as a function of time.



P4.147, P4.148

4.148 A rocket sled is to be slowed from an initial speed of 300 m/s by lowering a scoop into a water trough. The scoop is 0.3 m wide; it deflects the water through 150°. The trough is 800 m long. The mass of the sled is 8000 kg. At the initial speed it experiences an aerodynamic drag force of 90 kN. The aerodynamic force is proportional to the square of the sled speed. It is desired to slow the sled to 100 m/s. Determine the depth D to which the scoop must be lowered into the water.

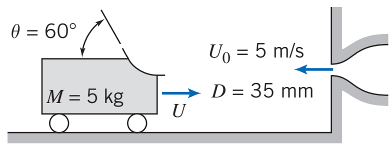
4.149 Starting from rest, the cart shown is propelled by a hydraulic catapult (liquid jet). The jet strikes the curved surface and makes a 180° turn, leaving horizontally. Air and rolling resistance may be neglected. If the mass of the cart is 100 kg and the jet of water leaves the nozzle (of area $0.001\ \text{m}^2$) with a speed of 35 m/s, determine the speed of the cart 5 s after the jet is directed against the cart. Plot the cart speed as a function of time.



P4.149, P4.150, P4.173

4.150 Consider the jet and cart of Problem 4.149 again, but include an aerodynamic drag force proportional to the square of cart speed, $F_D = kU^2$, with $k = 2.0\ \text{N} \cdot \text{s}^2/\text{m}^2$. Derive an expression for the cart acceleration as a function of cart speed and other given parameters. Evaluate the acceleration of the cart at $U = 10$ m/s. What fraction is this speed of the terminal speed of the cart?

4.151 A small cart that carries a single turning vane rolls on a level track. The cart mass is $M = 5$ kg and its initial speed is $U_0 = 5$ m/s. At $t = 0$, the vane is struck by an opposing jet of water, as shown. Neglect any external forces due to air or rolling resistance. Determine the jet speed V required to bring the cart to rest in (a) 1 s and (b) 2 s. In each case find the total distance traveled.

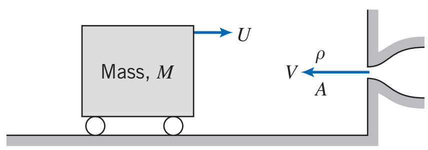


P4.151

4.152 Solve Problem 4.141 if the vane and slider ride on a film of oil instead of sliding in contact with the surface. Assume motion resistance is proportional to speed, $F_R = kU$, with $k = 7.5\ \text{N} \cdot \text{s/m}$.

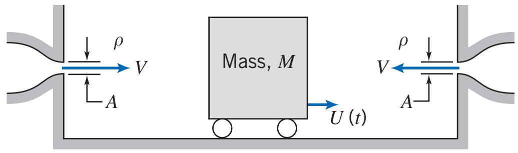
4.153 For the vane/slider problem of Problem 4.152, plot the acceleration, speed, and position of the slider as functions of time. (Consider numerical integration.)

4.154 A rectangular block of mass M, with vertical faces, rolls without resistance along a smooth horizontal plane as shown. The block travels initially at speed U_0. At $t = 0$ the block is struck by a liquid jet and its speed begins to slow. Obtain an algebraic expression for the acceleration of the block for $t > 0$. Solve the equation to determine the time at which $U = 0$.



P4.154, P4.156

4.155 A rectangular block of mass M, with vertical faces, rolls on a horizontal surface between two opposing jets as shown. At $t = 0$ the block is set into motion at speed U_0. Subsequently, it moves without friction parallel to the jet axes with speed $U(t)$. Neglect the mass of any liquid adhering to the block compared with M. Obtain general expressions for the acceleration of the block, $a(t)$, and the block speed, $U(t)$.

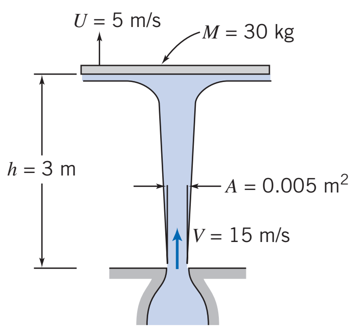


P4.155, P4.157

4.156 Consider the diagram of Problem 4.154. If $M = 100$ kg, $\rho = 999$ kg/m^3, and $A = 0.01$ m^2, find the jet speed V required for the cart to be brought to rest after one second if the initial speed of the cart is $U_0 = 5$ m/s. For this condition, plot the speed U and position x of the cart as functions of time. What is the maximum value of x, and how long does the cart take to return to its initial position?

4.157 Consider the statement and diagram of Problem 4.155. Assume that at $t = 0$, when the block of mass $M = 5$ kg is at $x = 0$, it is set into motion at speed $U_0 = 10$ m/s, to the right. The water jets have speed $V = 20$ m/s and area $A = 100$ mm^2. Calculate the time required to reduce the block speed to $U = 2.5$ m/s. Plot the block position x versus time. Compute the final rest position of the block. Explain why it comes to rest.

***4.158** A vertical jet of water impinges on a horizontal disk as shown. The disk assembly mass is 30 kg. When the disk is 3 m above the nozzle exit, it is moving upward at $U = 5$ m/s. Compute the vertical acceleration of the disk at this instant.



P4.158, P4.159, P4.180

4.159 A vertical jet of water leaves a 75-mm diameter nozzle. The jet impinges on a horizontal disk (see Problem 4.158). The disk is constrained horizontally but is free to move vertically. The mass of the disk is 35 kg. Plot disk mass versus flow rate to determine the water flow rate required to suspend the disk 3 m above the jet exit plane.

4.160 A rocket sled traveling on a horizontal track is slowed by a retro-rocket fired in the direction of travel. The initial speed of the sled is $U_0 = 500$ m/s. The initial mass of the sled is $M_0 = 1500$ kg. The retro-rocket consumes fuel at the rate of 7.75 kg/s, and the exhaust gases leave the nozzle at atmospheric pressure and a speed of 2500 m/s relative to the rocket. The retro-rocket fires for 20 s. Neglect aerodynamic drag and rolling resistance. Obtain and plot an algebraic expression for sled speed U as a function of firing time. Calculate the sled speed at the end of retro-rocket firing.

4.161 A manned space capsule travels in level flight above the Earth's atmosphere at initial speed $U_0 = 8.00$ km/s. The capsule is to be slowed by a retro-rocket to $U = 5.00$ km/s in preparation for a reentry maneuver. The initial mass of the capsule is $M_0 = 1600$ kg. The rocket consumes fuel at $\dot{m} = 8.0$ kg/s, and exhaust gases leave at $V_e = 3000$ m/s relative to the capsule and at negligible pressure. Evaluate the duration of the retro-rocket firing needed to accomplish this. Plot the final speed as a function of firing duration for a time range $\pm 10\%$ of this firing time.

4.162 A rocket sled accelerates from rest on a level track with negligible air and rolling resistances. The initial mass of the sled is $M_0 = 600$ kg. The rocket initially contains 150 kg of fuel. The rocket motor burns fuel at constant rate $\dot{m} = 15$ kg/s. Exhaust gases leave the rocket nozzle uniformly and axially at $V_e = 2900$ m/s relative to the nozzle, and the pressure is atmospheric. Find the maximum speed reached by the rocket sled. Calculate the maximum acceleration of the sled during the run.

4.163 A rocket sled has mass of 5000 kg, including 1000 kg of fuel. The motion resistance in the track on which the sled rides and that of the air total kU, where k is 50 N·s/m and U is the speed of the sled in m/s. The exit speed of the exhaust gas relative to the rocket is 1750 m/s, and the exit pressure is atmospheric. The rocket burns fuel at the rate of 50 kg/s.

(a) Plot the sled speed as a function of time.
(b) Find the maximum speed.
(c) What percentage increase in maximum speed would be obtained by reducing k by 10 percent?

4.164 A rocket sled with initial mass of 900 kg is to be accelerated on a level track. The rocket motor burns fuel at constant rate $\dot{m} = 13.5$ kg/s. The rocket exhaust flow is uniform and axial. Gases leave the nozzle at 2750 m/s relative to the nozzle, and the pressure is atmospheric. Determine the minimum mass of rocket fuel needed to propel the sled to a speed of 265 m/s before burnout occurs. As a first approximation, neglect resistance forces.

4.165 A rocket motor is used to accelerate a kinetic energy weapon to a speed of 3500 mph in horizontal flight. The exit stream leaves the nozzle axially and at atmospheric pressure with a speed of 6000 mph relative to the rocket. The rocket motor ignites upon release of the weapon from an aircraft flying horizontally at $U_0 = 600$ mph. Neglecting air resistance, obtain an algebraic expression for the speed reached by the weapon in level flight. Determine the minimum fraction of the initial mass of the weapon that must be fuel to accomplish the desired acceleration.

4.166 A rocket sled with initial mass of 3 metric tons, including 1 ton of fuel, rests on a level section of track. At $t = 0$, the solid fuel of the rocket is ignited and the rocket burns fuel at the rate of 75 kg/s. The exit speed of the exhaust gas relative to the rocket is 2500 m/s, and the pressure is atmospheric. Neglecting friction and air resistance, calculate the acceleration and speed of the sled at $t = 10$ s.

4.167 A daredevil considering a record attempt—for the world's longest motorcycle jump—asks for your consulting

*These problems require material from sections that may be omitted without loss of continuity in the text material.

help: He must reach 875 km/hr (from a standing start on horizontal ground) to make the jump, so he needs rocket propulsion. The total mass of the motorcycle, the rocket motor without fuel, and the rider is 375 kg. Gases leave the rocket nozzle horizontally, at atmospheric pressure, with a speed of 2510 m/s. Evaluate the minimum amount of rocket fuel needed to accelerate the motorcycle and rider to the required speed.

4.168 A "home-made" solid propellant rocket has an initial mass of 20 lbm; 15 lbm of this is fuel. The rocket is directed vertically upward from rest, burns fuel at a constant rate of 0.5 lbm/s, and ejects exhaust gas at a speed of 6500 ft/s relative to the rocket. Assume that the pressure at the exit is atmospheric and that air resistance may be neglected. Calculate the rocket speed after 20 s and the distance traveled by the rocket in 20 s. Plot the rocket speed and the distance traveled as functions of time.

4.169 A large two-stage liquid rocket with mass of 30,000 kg is to be launched from a sea-level launch pad. The main engine burns liquid hydrogen and liquid oxygen in a stoichiometric mixture at 2450 kg/s. The thrust nozzle has an exit diameter of 2.6 m. The exhaust gases exit the nozzle at 2270 m/s and an exit plane pressure of 66 kPa absolute. Calculate the acceleration of the rocket at liftoff. Obtain an expression for speed as a function of time, neglecting air resistance.

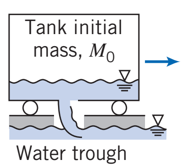
4.170 Neglecting air resistance, what speed would a vertically directed rocket attain in 5 s if it starts from rest, has initial mass of 350 kg, burns 10 kg/s, and ejects gas at atmospheric pressure with a speed of 2500 m/s relative to the rocket? What would be the maximum velocity? Plot the rocket speed as a function of time for the first minute of flight.

4.171 Inflate a toy balloon with air and release it. Watch as the balloon darts about the room. Explain what causes the phenomenon you see.

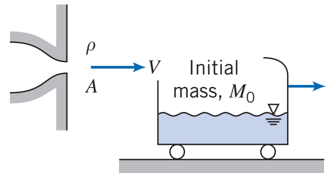
4.172 The vane/cart assembly of mass $M = 30$ kg, shown in Problem 4.128, is driven by a water jet. The water leaves the stationary nozzle of area $A = 0.02\ \text{m}^2$, with a speed of 20 m/s. The coefficient of kinetic friction between the assembly and the surface is 0.10. Plot the terminal speed of the assembly as a function of vane turning angle, θ, for $0 \le \theta \le \pi/2$. At what angle does the assembly begin to move if the coefficient of static friction is 0.15?

4.173 Consider the vehicle shown in Problem 4.149. Starting from rest, it is propelled by a hydraulic catapult (liquid jet). The jet strikes the curved surface and makes a 180° turn, leaving horizontally. Air and rolling resistance may be neglected. Using the notation shown, obtain an equation for the acceleration of the vehicle at any time and determine the time required for the vehicle to reach $U = V/2$.

4.174 The moving tank shown is to be slowed by lowering a scoop to pick up water from a trough. The initial mass and speed of the tank and its contents are M_0 and U_0, respectively. Neglect external forces due to pressure or friction and assume that the track is horizontal. Apply the continuity and momentum equations to show that at any instant $U = U_0 M_0/M$. Obtain a general expression for U/U_0 as a function of time.



P4.174



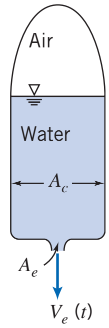
P4.175

4.175 The tank shown rolls with negligible resistance along a horizontal track. It is to be accelerated from rest by a liquid jet that strikes the vane and is deflected into the tank. The initial mass of the tank is M_0. Use the continuity and momentum equations to show that at any instant the mass of the vehicle and liquid contents is $M = M_0 V/(V - U)$. Obtain a general expression for U/V as a function of time.

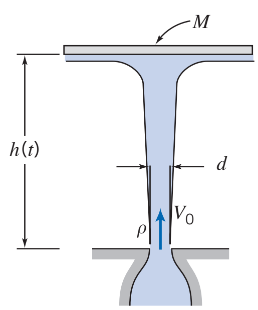
4.176 A model solid propellant rocket has a mass of 69.6 g, of which 12.5 g is fuel. The rocket produces 5.75 N of thrust for a duration of 1.7 s. For these conditions, calculate the maximum speed and height attainable in the absence of air resistance. Plot the rocket speed and the distance traveled as functions of time.

4.177 A small rocket motor is used to power a "jet pack" device to lift a single astronaut above the Moon's surface. The rocket motor produces a uniform exhaust jet with constant speed, $V_e = 3000$ m/s, and the thrust is varied by changing the jet size. The total initial mass of the astronaut and the jet pack is $M_0 = 200$ kg, 100 kg of which is fuel and oxygen for the rocket motor. Find (a) the exhaust mass flow rate required to just lift off initially, (b) the mass flow rate just as the fuel and oxygen are used up, and (c) the maximum anticipated time of flight. Note that the Moon's gravity is about 17 percent of Earth's.

***4.178** Several toy manufacturers sell water "rockets" that consist of plastic tanks to be partially filled with water and then pressurized with air. Upon release, the compressed air forces water out of the nozzle rapidly, propelling the rocket. You are asked to help specify optimum conditions for this water-jet propulsion system. To simplify the analysis, consider horizontal motion only. Perform the analysis and design needed to define the acceleration performance of the compressed air/water-propelled rocket. Identify the fraction of tank volume that initially should be filled with compressed air to achieve optimum performance (i.e., maximum speed from the water charge). Describe the effect of varying the initial air pressure in the tank.



P4.178



P4.179

*These problems require material from sections that may be omitted without loss of continuity in the text material.

***4.179** A disk, of mass M, is constrained horizontally but is free to move vertically. A jet of water strikes the disk from below. The jet leaves the nozzle at initial speed V_0. Obtain a differential equation for the disk height, $h(t)$, above the jet exit plane if the disk is released from large height, H. (You will not be able to solve this ODE, as it is highly nonlinear!) Assume that when the disk reaches equilibrium, its height above the jet exit plane is h_0.
(a) Sketch $h(t)$ for the disk released at $t = 0$ from $H > h_0$.
(b) Explain why the sketch is as you show it.

***4.180** Consider the configuration of the vertical jet impinging on a horizontal disk shown in Problem 4.158. Assume the disk is released from rest at an initial height of 2 m above the jet exit plane. Using a numerical method such as the Euler method (see Section 5.5), solve for the subsequent motion of this disk. Identify the steady-state height of the disk.

4.181 A small solid-fuel rocket motor is fired on a test stand. The combustion chamber is circular, with 100 mm diameter. Fuel, of density 1660 kg/m^3, burns uniformly at the rate of 12.7 mm/s. Measurements show that the exhaust gases leave the rocket at ambient pressure, at a speed of 2750 m/s. The absolute pressure and temperature in the combustion chamber are 7.0 MPa and 3610 K, respectively. Treat the combustion products as an ideal gas with molecular mass of 25.8. Evaluate the rate of change of mass and of linear momentum within the rocket motor. Express the rate of change of linear momentum within the motor as a percentage of the motor thrust.

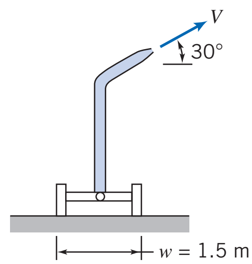
***4.182** The capability of the Aircraft Landing Loads and Traction Facility at NASA's Langley Research Center is to be upgraded. The facility consists of a rail-mounted carriage propelled by a jet of water issuing from a pressurized tank. (The setup is identical in concept to the hydraulic catapult of Problem 4.138.) Specifications require accelerating the carriage with 49,000 kg mass to a speed of 220 knots in a distance of 122 m. (The vane turning angle is 170°.) Identify a range of water jet sizes and speeds needed to accomplish this performance. Specify the recommended operating pressure for the water-jet system and determine the shape and estimated size of tankage to contain the pressurized water.

***4.183** A classroom demonstration of linear momentum is planned, using a water-jet propulsion system for a cart traveling on a horizontal linear air track. The track is 5 m long, and the cart mass is 155 g. The objective of the design is to obtain the best performance for the cart, using 1 L of water contained in an open cylindrical tank made from plastic sheet with density of 0.0819 g/cm^2. For stability, the maximum height of the water tank cannot exceed 0.5 m. The diameter of the smoothly rounded water jet may not exceed 10 percent of the tank diameter. Determine the best dimensions for the tank and the water jet by modeling the system performance. Using a numerical method such as the Euler method (see Section 5.5), plot acceleration, velocity, and distance as functions of time. Find the optimum dimensions of the water tank and jet opening from the tank. Discuss the limitations on your analysis. Discuss how the assumptions affect the predicted performance of the cart. Would the actual performance of the cart be better or worse than predicted? Why? What factors account for the difference(s)?

***4.184** Analyze the design and optimize the performance of a cart propelled along a horizontal track by a water jet that issues under gravity from an open cylindrical tank carried on board the cart. (A water-jet-propelled cart is shown in the diagram for Problem 4.142.) Neglect any change in slope of the liquid free surface in the tank during acceleration. Analyze the motion of the cart along a horizontal track, assuming it starts from rest and begins to accelerate when water starts to flow from the jet. Derive algebraic equations or solve numerically for the acceleration and speed of the cart as functions of time. Present results as plots of acceleration and speed versus time, neglecting the mass of the tank. Determine the dimensions of a tank of minimum mass required to accelerate the cart from rest along a horizontal track to a specified speed in a specified time interval.

The Angular-Momentum Principle

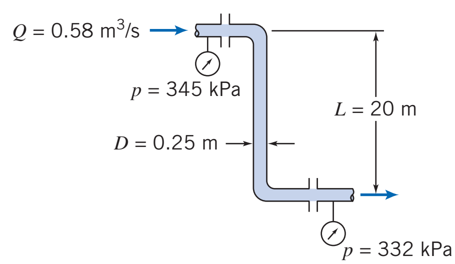
***4.185** A large irrigation sprinkler unit, mounted on a cart, discharges water with a speed of 40 m/s at an angle of 30° to the horizontal. The 50-mm-diameter nozzle is 3 m above the ground. The mass of the sprinkler and cart is $M = 350$ kg. Calculate the magnitude of the moment that tends to overturn the cart. What value of V will cause impending motion? What will be the nature of the impending motion? What is the effect of the angle of jet inclination on the results? For the case of impending motion, plot the jet velocity as a function of the angle of jet inclination over an appropriate range of the angles.



P4.185

***4.186** The 90° reducing elbow of Example 4.6 discharges to atmosphere. Section ② is located 0.3 m to the right of Section ①. Estimate the moment exerted by the flange on the elbow.

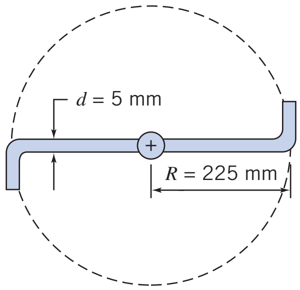
***4.187** Crude oil (SG = 0.95) from a tanker dock flows through a pipe of 0.25 m diameter in the configuration shown. The flow rate is 0.58 m^3/s, and the gage pressures are shown in the diagram. Determine the force and torque that are exerted by the pipe assembly on its supports.



P4.187

*These problems require material from sections that may be omitted without loss of continuity in the text material.

***4.188** The simplified lawn sprinkler shown rotates in the horizontal plane. At the center pivot, $Q = 15$ L/min of water enters vertically. Water discharges in the horizontal plane from each jet. If the pivot is frictionless, calculate the torque needed to keep the sprinkler from rotating. Neglecting the inertia of the sprinkler itself, calculate the angular acceleration that results when the torque is removed.

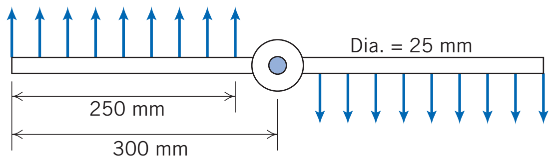


P4.188, P4.189, P4.190

***4.189** Consider the sprinkler of Problem 4.188 again. Derive a differential equation for the angular speed of the sprinkler as a function of time. Evaluate its steady-state speed of rotation if there is no friction in the pivot.

***4.190** Repeat Problem 4.189, but assume a constant retarding torque in the pivot of 0.5 N·m. At what retarding torque would the sprinkler not be able to rotate?

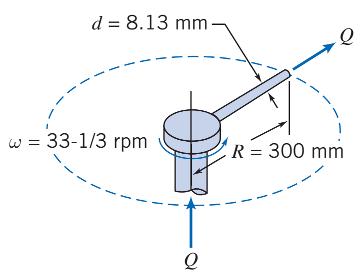
***4.191** Water flows in a uniform flow out of the 2.5-mm slots of the rotating spray system, as shown. The flow rate is 3 L/s. Find (a) the torque required to hold the system stationary and (b) the steady-state speed of rotation after it is released.



P4.191, P4.192

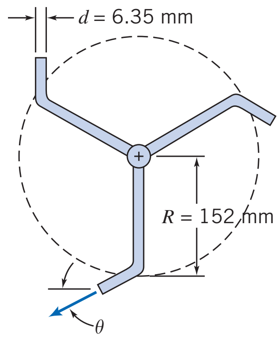
***4.192** If the same flow rate in the rotating spray system of Problem 4.191 is not uniform but instead varies linearly from a maximum at the outer radius to zero at the inner radius, find (a) the torque required to hold it stationary and (b) the steady-state speed of rotation.

***4.193** A single tube carrying water rotates at constant angular speed, as shown. Water is pumped through the tube at volume flow rate $Q = 13.8$ L/min. Find the torque that must be applied to maintain the steady rotation of the tube using two methods of analysis: (a) a rotating control volume and (b) a fixed control volume.



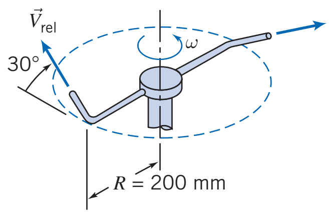
P4.193

***4.194** The lawn sprinkler shown is supplied with water at a rate of 68 L/min. Neglecting friction in the pivot, determine the steady-state angular speed for $\theta = 30°$. Plot the steady-state angular speed of the sprinkler for $0 \leq \theta \leq 90°$.



P4.194

***4.195** A small lawn sprinkler is shown. The sprinkler operates at a gage pressure of 140 kPa. The total flow rate of water through the sprinkler is 4 L/min. Each jet discharges at 17 m/s (relative to the sprinkler arm) in a direction inclined 30° above the horizontal. The sprinkler rotates about a vertical axis. Friction in the bearing causes a torque of 0.18 N · m opposing rotation. Evaluate the torque required to hold the sprinkler stationary.



P4.195, P4.196, P4.197

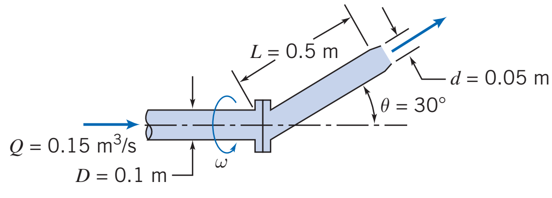
***4.196** In Problem 4.195, calculate the initial acceleration of the sprinkler from rest if no external torque is applied and the moment of inertia of the sprinkler head is 0.1 kg · m^2 when filled with water.

***4.197** A small lawn sprinkler is shown (Problem 4.196). The sprinkler operates at an inlet gage pressure of 140 kPa. The total flow rate of water through the sprinkler is 4.0 L/min. Each jet discharges at 17 m/s (relative to the sprinkler arm) in a direction inclined 30° above the horizontal. The sprinkler rotates about a vertical axis. Friction in the bearing causes a torque of 0.18 N · m opposing rotation. Determine the steady speed of rotation of the sprinkler and the approximate area covered by the spray.

***4.198** When a garden hose is used to fill a bucket, water in the bucket may develop a swirling motion. Why does this happen? How could the amount of swirl be calculated approximately?

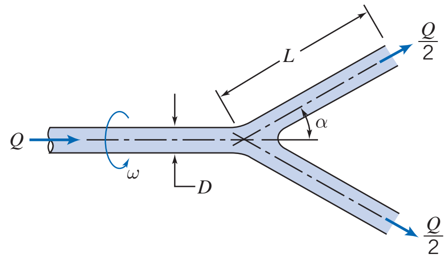
***4.199** Water flows at the rate of 0.15 m^3/s through a nozzle assembly that rotates steadily at 30 rpm. The arm and nozzle masses are negligible compared with the water inside. Determine the torque required to drive the device and the reaction torques at the flange.

*These problems require material from sections that may be omitted without loss of continuity in the text material.



P4.199

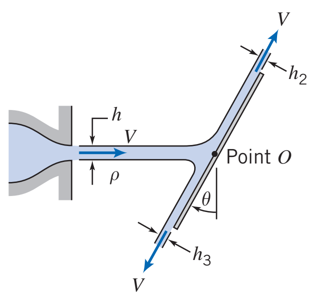
*4.200 A pipe branches symmetrically into two legs of length L, and the whole system rotates with angular speed ω around its axis of symmetry. Each branch is inclined at angle α to the axis of rotation. Liquid enters the pipe steadily, with zero angular momentum, at volume flow rate Q. The pipe diameter, D, is much smaller than L. Obtain an expression for the external torque required to turn the pipe. What additional torque would be required to impart angular acceleration $\dot{\omega}$?



P4.200

*4.201 Liquid in a thin sheet, of width w and thickness h, flows from a slot and strikes a stationary inclined flat plate, as shown. Experiments show that the resultant force of the liquid jet on the plate does not act through point O, where the jet centerline intersects the plate. Determine the magnitude and line of application of the resultant force as functions of θ. Evaluate the equilibrium angle of the plate if the resultant force is applied at point O. Neglect any viscous effects.

*4.202 For the rotating sprinkler of Example 4.14, what value of α will produce the maximum rotational speed? What angle will provide the maximum area of coverage by the spray? Draw a velocity diagram (using an r, θ, z coordinate system) to indicate the absolute velocity of the water jet leaving the nozzle. What governs the steady rotational speed of the sprinkler? Does the rotational speed of the sprinkler affect the area covered by the spray? How would you estimate the area? For fixed α, what might be done to increase or decrease the area covered by the spray?



P4.201

The First Law of Thermodynamics

4.203 Air at standard conditions enters a compressor at 75 m/s and leaves at an absolute pressure and temperature of 200 kPa and 345 K, respectively, and speed $V = 125$ m/s. The flow rate is 1 kg/s. The cooling water circulating around the compressor casing removes 18 kJ/kg of air. Determine the power required by the compressor.

4.204 Compressed air is stored in a pressure bottle with a volume of 100 L, at 500 kPa and 20°C. At a certain instant, a valve is opened and mass flows from the bottle at $\dot{m} = 0.01$ kg/s. Find the rate of change of temperature in the bottle at this instant

4.205 A centrifugal water pump with a 0.1-m-diameter inlet and a 0.1-m-diameter discharge pipe has a flow rate of 0.02 m^3/s. The inlet pressure is 0.2 m Hg vacuum and the exit pressure is 240 kPa. The inlet and outlet sections are located at the same elevation. The measured power input is 6.75 kW. Determine the pump efficiency.

4.206 A turbine is supplied with 0.6 m^3/s of water from a 0.3-m-diameter pipe; the discharge pipe has a 0.4 m diameter. Determine the pressure drop across the turbine if it delivers 60 kW.

4.207 Air enters a compressor at 14 psia, 80°F with negligible speed and is discharged at 70 psia, 500°F with a speed of 500 ft/s. If the power input is 3200 hp and the flow rate is 20 lbm/s, determine the rate of heat transfer.

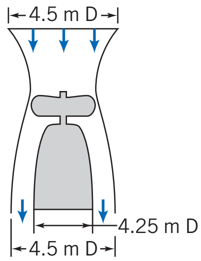
4.208 Air is drawn from the atmosphere into a turbomachine. At the exit, conditions are 500 kPa (gage) and 130°C. The exit speed is 100 m/s and the mass flow rate is 0.8 kg/s. Flow is steady and there is no heat transfer. Compute the shaft work interaction with the surroundings.

4.209 All major harbors are equipped with fire boats for extinguishing ship fires. A 3-in.-diameter hose is attached to the discharge of a 15-hp pump on such a boat. The nozzle attached to the end of the hose has a diameter of 1 in. If the nozzle discharge is held 10 ft above the surface of the water, determine the volume flow rate through the nozzle, the maximum height to which the water will rise, and the force on the boat if the water jet is directed horizontally over the stern.

4.210 A pump draws water from a reservoir through a 150-mm-diameter suction pipe and delivers it to a 75-mm-diameter discharge pipe. The end of the suction pipe is 2 m below the free surface of the reservoir. The pressure gage on the discharge pipe (2 m above the reservoir surface) reads 170 kPa. The average speed in the discharge pipe is 3 m/s. If the pump efficiency is 75 percent, determine the power required to drive it.

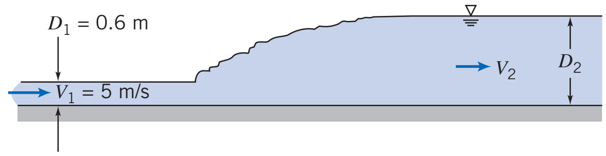
4.211 The total mass of the helicopter-type craft shown is 1000 kg. The pressure of the air is atmospheric at the outlet. Assume the flow is steady and one-dimensional. Treat the air as incompressible at standard conditions and calculate, for a hovering position, the speed of the air leaving the craft and the minimum power that must be delivered to the air by the propeller.

*These problems require material from sections that may be omitted without loss of continuity in the text material.



P4.211

4.212 Liquid flowing at high speed in a wide, horizontal open channel under some conditions can undergo a hydraulic jump, as shown. For a suitably chosen control volume, the flows entering and leaving the jump may be considered uniform with hydrostatic pressure distributions (see Example 4.7). Consider a channel of width w, with water flow at $D_1 = 0.6$ m and $V_1 = 5$ m/s. Show that in general, $D_2 = D_1\left[\sqrt{1+8V_1^2/gD_1}-1\right]/2$.



P4.212

Evaluate the change in mechanical energy through the hydraulic jump. If heat transfer to the surroundings is negligible, determine the change in water temperature through the jump.

5

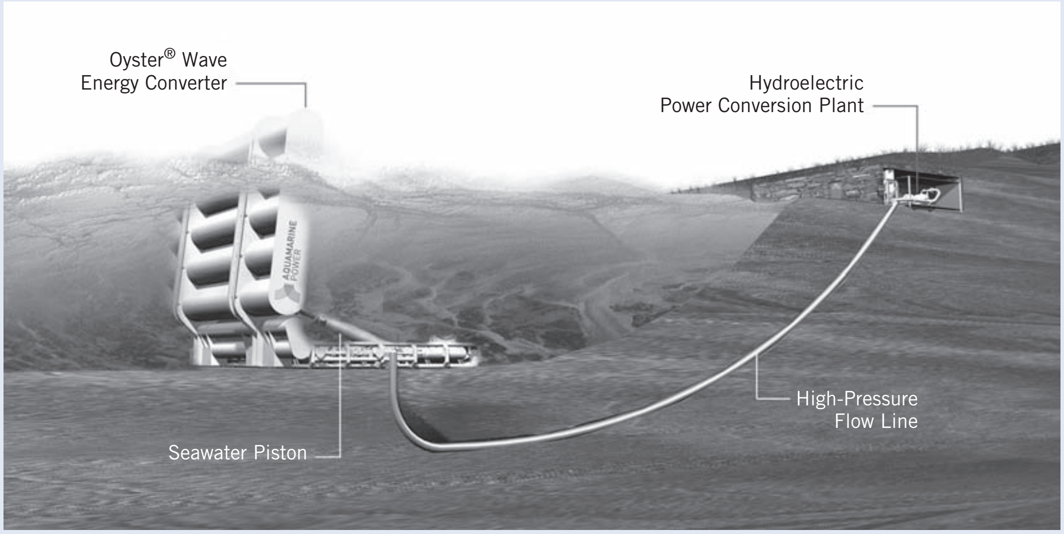
Introduction to Differential Analysis of Fluid Motion

Case Study in Energy and the Environment

Wave Power: *Aquamarine Oyster Wave Energy Converter*

Aquamarine Power, a wave energy company located in Scotland, has developed an innovative hydroelectric wave energy converter, known as *Oyster*; a demonstration-scale model was installed in 2009 and began producing power for homes in some regions of Scotland. They eventually plan to have commercially viable *Oyster* wave farms around the world, the first planned for 2013. A farm of 20 *Oyster* devices would provide enough energy to power 9000 homes, offsetting carbon emissions of about 20,000 metric tons.

The *Oyster* device consists of a simple mechanical hinged flap, as shown in the figure, connected to the seabed at around a 10-m depth. As each wave passes by, it forces the flap to move; the flap in turn drives



A schematic of Aquamarine's Oyster device (Picture courtesy of Aquamarine Power)

hydraulic pistons to deliver high-pressure water, via a pipeline, to an onshore electrical turbine. *Oyster* farms using multiple devices are expected to be capable of generating 100 MW or more.

Oyster has a number of advantages: It has good efficiency and durability, and, with its low-cost operation, maintenance, and manufacture, it is hoped it will produce reliable *cost-competitive* electricity from the waves for the first time. The device uses simple and robust mechanical offshore component, combined with proven conventional onshore hydroelectric components. Designed with the notion that simple is best, less is more, it has a minimum of offshore submerged moving parts; there are no underwater generators, power electronics, or gearboxes. The *Oyster* is designed to take advantage of the more consistent waves found near the shore; for durability, any excess energy from exceptionally large waves simply spills over the top of *Oyster*'s flap. Its motion allows it to literally duck under such waves. *Aquamarine Power* believes its device is competitive with devices weighing up to five times as much, and, with multiple pumps feeding a single onshore generator, *Oyster* will offer good economies of scale. As a final bonus, *Oyster* uses water instead of oil as its hydraulic fluid for minimum environmental impact and generates essentially no noise pollution.

In Chapter 4, we developed the basic equations in integral form for a control volume. Integral equations are useful when we are interested in the gross behavior of a flow field and its effect on various devices. However, the integral approach does not enable us to obtain detailed point-by-point knowledge of the flow field. For example, the integral approach could provide information on the lift generated by a wing; it could not be used to determine the pressure distribution that produced the lift on the wing.

To see what is happening in a flow in detail, we need differential forms of the equations of motion. In this chapter we shall develop differential equations for the conservation of mass and Newton's second law of motion. Since we are interested in developing differential equations, we will need to analyze infinitesimal systems and control volumes.

5.1 Conservation of Mass

In Chapter 2, we developed the field representation of fluid properties. The property fields are defined by continuous functions of the space coordinates and time. The density and velocity fields were related through conservation of mass in integral form in Chapter 4 (Eq. 4.12). In this chapter we shall derive the differential equation for

conservation of mass in rectangular and in cylindrical coordinates. In both cases the derivation is carried out by applying conservation of mass to a differential control volume.

Rectangular Coordinate System

In rectangular coordinates, the control volume chosen is an infinitesimal cube with sides of length dx, dy, dz as shown in Fig. 5.1. The density at the center, O, of the control volume is assumed to be ρ and the velocity there is assumed to be $\vec{V} = \hat{i}u + \hat{j}v + \hat{k}w$.

To evaluate the properties at each of the six faces of the control surface, we use a Taylor series expansion about point O. For example, at the right face,

$$\rho)_{x+dx/2} = \rho + \left(\frac{\partial \rho}{\partial x}\right)\frac{dx}{2} + \left(\frac{\partial^2 \rho}{\partial x^2}\right)\frac{1}{2!}\left(\frac{dx}{2}\right)^2 + \cdots$$

Neglecting higher-order terms, we can write

$$\rho)_{x+dx/2} = \rho + \left(\frac{\partial \rho}{\partial x}\right)\frac{dx}{2}$$

and

$$u)_{x+dx/2} = u + \left(\frac{\partial u}{\partial x}\right)\frac{dx}{2}$$

where ρ, u, $\partial\rho/\partial x$, and $\partial u/\partial x$ are all evaluated at point O. The corresponding terms at the left face are

$$\rho)_{x-dx/2} = \rho + \left(\frac{\partial \rho}{\partial x}\right)\left(-\frac{dx}{2}\right) = \rho - \left(\frac{\partial \rho}{\partial x}\right)\frac{dx}{2}$$

$$u)_{x-dx/2} = u + \left(\frac{\partial u}{\partial x}\right)\left(-\frac{dx}{2}\right) = u - \left(\frac{\partial u}{\partial x}\right)\frac{dx}{2}$$

We can write similar expressions involving ρ and v for the front and back faces and ρ and w for the top and bottom faces of the infinitesimal cube $dx\,dy\,dz$. These can then be used to evaluate the surface integral in Eq. 4.12 (recall that $\int_{CS} \rho\vec{V}\cdot d\vec{A}$ is the net flux of mass out of the control volume):

$$\frac{\partial}{\partial t}\int_{CV} \rho\, d\forall + \int_{CS} \rho\vec{V}\cdot d\vec{A} = 0 \tag{4.12}$$

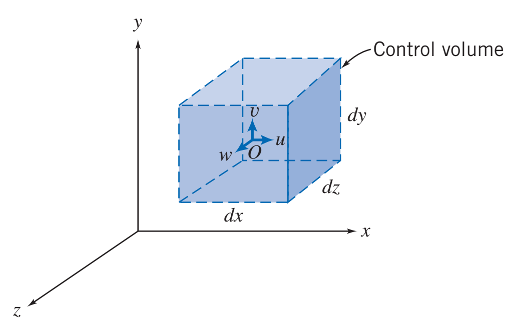


Fig. 5.1 Differential control volume in rectangular coordinates.

Table 5.1

Mass Flux Through the Control Surface of a Rectangular Differential Control Volume

Surface	Evaluation of $\int \rho\vec{V}\cdot d\vec{A}$
Left $(-x)$	$= -\left[\rho - \left(\frac{\partial \rho}{\partial x}\right)\frac{dx}{2}\right]\left[u - \left(\frac{\partial u}{\partial x}\right)\frac{dx}{2}\right]dy\,dz = -\rho u\,dy\,dz + \frac{1}{2}\left[u\left(\frac{\partial \rho}{\partial x}\right) + \rho\left(\frac{\partial u}{\partial x}\right)\right]dx\,dy\,dz$
Right $(+x)$	$= \left[\rho + \left(\frac{\partial \rho}{\partial x}\right)\frac{dx}{2}\right]\left[u + \left(\frac{\partial u}{\partial x}\right)\frac{dx}{2}\right]dy\,dz = \rho u\,dy\,dz + \frac{1}{2}\left[u\left(\frac{\partial \rho}{\partial x}\right) + \rho\left(\frac{\partial u}{\partial x}\right)\right]dx\,dy\,dz$
Bottom $(-y)$	$= -\left[\rho - \left(\frac{\partial \rho}{\partial y}\right)\frac{dy}{2}\right]\left[v - \left(\frac{\partial v}{\partial y}\right)\frac{dy}{2}\right]dx\,dz = -\rho v\,dx\,dz + \frac{1}{2}\left[v\left(\frac{\partial \rho}{\partial y}\right) + \rho\left(\frac{\partial v}{\partial y}\right)\right]dx\,dy\,dz$
Top $(+y)$	$= \left[\rho + \left(\frac{\partial \rho}{\partial y}\right)\frac{dy}{2}\right]\left[v + \left(\frac{\partial v}{\partial y}\right)\frac{dy}{2}\right]dx\,dz = \rho v\,dx\,dz + \frac{1}{2}\left[v\left(\frac{\partial \rho}{\partial y}\right) + \rho\left(\frac{\partial v}{\partial y}\right)\right]dx\,dy\,dz$
Back $(-z)$	$= -\left[\rho - \left(\frac{\partial \rho}{\partial z}\right)\frac{dz}{2}\right]\left[w - \left(\frac{\partial w}{\partial z}\right)\frac{dz}{2}\right]dx\,dy = -\rho w\,dx\,dy + \frac{1}{2}\left[w\left(\frac{\partial \rho}{\partial z}\right) + \rho\left(\frac{\partial w}{\partial z}\right)\right]dx\,dy\,dz$
Front $(+z)$	$= \left[\rho + \left(\frac{\partial \rho}{\partial z}\right)\frac{dz}{2}\right]\left[w + \left(\frac{\partial w}{\partial z}\right)\frac{dz}{2}\right]dx\,dy = \rho w\,dx\,dy + \frac{1}{2}\left[w\left(\frac{\partial \rho}{\partial z}\right) + \rho\left(\frac{\partial w}{\partial z}\right)\right]dx\,dy\,dz$

Adding the results for all six faces,

$$\int_{CS} \rho\vec{V}\cdot d\vec{A} = \left[\left\{u\left(\frac{\partial \rho}{\partial x}\right) + \rho\left(\frac{\partial u}{\partial x}\right)\right\} + \left\{v\left(\frac{\partial \rho}{\partial y}\right) + \rho\left(\frac{\partial v}{\partial y}\right)\right\} + \left\{w\left(\frac{\partial \rho}{\partial z}\right) + \rho\left(\frac{\partial w}{\partial z}\right)\right\}\right]dx\,dy\,dz$$

or

$$\int_{CS} \rho\vec{V}\cdot d\vec{A} = \left[\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z}\right]dx\,dy\,dz$$

Table 5.1 shows the details of this evaluation. Note: We assume that the velocity components u, v, and w are positive in the x, y, and z directions, respectively; the area normal is by convention positive out of the cube; and higher-order terms [e.g., $(dx)^2$] are neglected in the limit as dx, dy, and $dz \rightarrow 0$.

The result of all this work is

$$\left[\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial x} + \frac{\partial \rho w}{\partial x}\right]dx\,dy\,dz$$

This expression is the surface integral evaluation for our differential cube. To complete Eq. 4.12, we need to evaluate the volume integral (recall that $\partial/\partial t \int_{CV} \rho d\forall$ is the rate of change of mass in the control volume):

$$\frac{\partial}{\partial t}\int_{CV} \rho d\forall \rightarrow \frac{\partial}{\partial t}[\rho dx\,dy\,dz] = \frac{\partial \rho}{\partial t}dx\,dy\,dz$$

Hence, we obtain (after canceling $dx\,dy\,dz$) from Eq. 4.12 a differential form of the mass conservation law

$$\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} + \frac{\partial \rho}{\partial t} = 0 \tag{5.1a}$$

Equation 5.1a is frequently called the *continuity equation*.

Since the vector operator, ∇, in rectangular coordinates, is given by

$$\nabla = \hat{i}\frac{\partial}{\partial x} + \hat{j}\frac{\partial}{\partial y} + \hat{k}\frac{\partial}{\partial z}$$

then

$$\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} = \nabla \cdot \rho \vec{V}$$

Note that the del operator ∇ acts on ρ *and* $\vec{V}$. Think of it as $\nabla \cdot (\rho \vec{V})$. The conservation of mass may be written as

$$\nabla \cdot \rho \vec{V} + \frac{\partial \rho}{\partial t} = 0 \tag{5.1b}$$

Two flow cases for which the differential continuity equation may be simplified are worthy of note.

For an *incompressible* fluid, $\rho = \text{constant}$; density is neither a function of space coordinates nor a function of time. For an incompressible fluid, the continuity equation simplifies to

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = \nabla \cdot \vec{V} = 0 \tag{5.1c}$$

Thus the velocity field, $\vec{V}(x, y, z, t)$, for incompressible flow must satisfy $\nabla \cdot \vec{V} = 0$.

For *steady* flow, all fluid properties are, by definition, independent of time. Thus $\partial \rho / \partial t = 0$ and at most $\rho = \rho(x, y, z)$. For steady flow, the continuity equation can be written as

$$\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} = \nabla \cdot \rho \vec{V} = 0 \tag{5.1d}$$

(and remember that the del operator ∇ acts on ρ *and* $\vec{V}$).

Example 5.1 INTEGRATION OF TWO-DIMENSIONAL DIFFERENTIAL CONTINUITY EQUATION

For a two-dimensional flow in the xy plane, the x component of velocity is given by $u = Ax$. Determine a possible y component for incompressible flow. How many y components are possible?

Given: Two-dimensional flow in the xy plane for which $u = Ax$.

Find: (a) Possible y component for incompressible flow.
(b) Number of possible y components.

Solution:

Governing equation: $\nabla \cdot \rho \vec{V} + \dfrac{\partial \rho}{\partial t} = 0$

For incompressible flow this simplifies to $\nabla \cdot \vec{V} = 0$. In rectangular coordinates

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

For two-dimensional flow in the xy plane, $\vec{V} = \vec{V}(x, y)$. Then partial derivatives with respect to z are zero, and

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

Then

$$\frac{\partial v}{\partial y} = -\frac{\partial u}{\partial x} = -A$$

which gives an expression for the rate of change of v holding x constant. This equation can be integrated to obtain an expression for v. The result is

$$v = \int \frac{\partial v}{\partial y} dy + f(x, t) = -Ay + f(x, t) \longleftarrow v$$

{The function of x and t appears because we had a partial derivative of v with respect to y.}

Any function $f(x, t)$ is allowable, since $\partial/\partial y \, f(x, t) = 0$. Thus any number of expressions for v could satisfy the differential continuity equation under the given conditions. The simplest expression for v would be obtained by setting $f(x, t) = 0$. Then $v = -Ay$, and

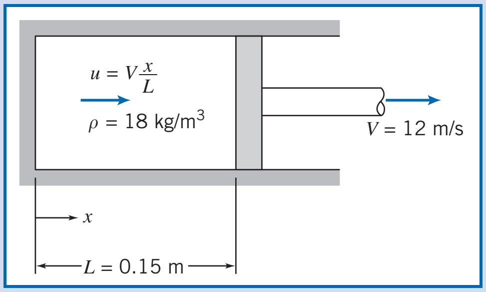
$$\vec{V} = Ax\hat{i} - Ay\hat{j} \longleftarrow \vec{V}$$

This problem:
- ✓ Shows use of the differential continuity equation for obtaining information on a flow field.
- ✓ Demonstrates integration of a partial derivative.
- ✓ Proves that the flow originally discussed in Example 2.1 is indeed incompressible.

Example 5.2 UNSTEADY DIFFERENTIAL CONTINUITY EQUATION

A gas-filled pneumatic strut in an automobile suspension system behaves like a piston-cylinder apparatus. At one instant when the piston is $L = 0.15$ m away from the closed end of the cylinder, the gas density is uniform at $\rho = 18$ kg/m^3 and the piston begins to move away from the closed end at $V = 12$ m/s. Assume as a simple model that the gas velocity is one-dimensional and proportional to distance from the closed end; it varies linearly from zero at the end to $u = V$ at the piston. Find the rate of change of gas density at this instant. Obtain an expression for the average density as a function of time.

Given: Piston-cylinder as shown.



Find: (a) Rate of change of density.
(b) $\rho(t)$.

Solution:

Governing equation: $\nabla \cdot \rho \vec{V} + \dfrac{\partial \rho}{\partial t} = 0$

In rectangular coordinates, $\dfrac{\partial \rho u}{\partial x} + \dfrac{\partial \rho v}{\partial y} + \dfrac{\partial \rho w}{\partial z} + \dfrac{\partial \rho}{\partial t} = 0$

Since $u = u(x)$, partial derivatives with respect to y and z are zero, and

$$\frac{\partial \rho u}{\partial x} + \frac{\partial \rho}{\partial t} = 0$$

Then

$$\frac{\partial \rho}{\partial t} = -\frac{\partial \rho u}{\partial x} = -\rho\frac{\partial u}{\partial x} - u\frac{\partial \rho}{\partial x}$$

Since ρ is assumed uniform in the volume, $\dfrac{\partial \rho}{\partial x} = 0$, and $\dfrac{\partial \rho}{\partial t} = \dfrac{d\rho}{dt} = -\rho\dfrac{\partial u}{\partial x}$.

Since $u = V\dfrac{x}{L}$, $\dfrac{\partial u}{\partial x} = \dfrac{V}{L}$, then $\dfrac{d\rho}{dt} = -\rho\dfrac{V}{L}$. However, note that $L = L_0 + Vt$.

Separate variables and integrate,

$$\int_{\rho_0}^{\rho} \frac{d\rho}{\rho} = -\int_0^t \frac{V}{L}\,dt = -\int_0^t \frac{V\,dt}{L_0 + Vt}$$

$$\ln\frac{\rho}{\rho_0} = \ln\frac{L_0}{L_0 + Vt} \quad \text{and} \quad \rho(t) = \rho_0\left[\frac{1}{1 + Vt/L_0}\right] \longleftarrow \rho(t)$$

At $t = 0$,

$$\frac{\partial \rho}{\partial t} = -\rho_0\frac{V}{L} = -18\,\frac{\text{kg}}{\text{m}^3} \times 12\,\frac{\text{m}}{\text{s}} \times \frac{1}{0.15\,\text{m}} = -1440\,\text{kg}/(\text{m}^3\cdot\text{s}) \longleftarrow \frac{\partial \rho}{\partial t}$$

This problem demonstrates use of the differential continuity equation for obtaining the density variation with time for an unsteady flow.

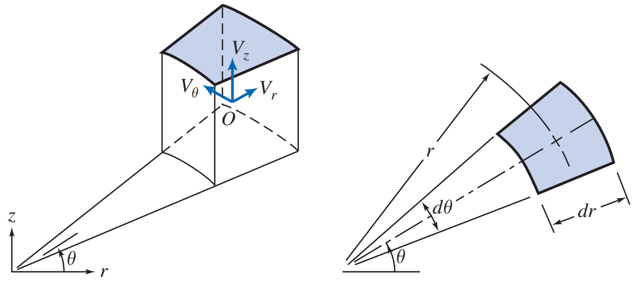
The density-time graph is shown in an *Excel* workbook. This workbook is interactive: It allows one to see the effect of different values of ρ_0, L, and V on ρ versus t. Also, the time at which the density falls to any prescribed value can be determined.

Cylindrical Coordinate System

A suitable differential control volume for cylindrical coordinates is shown in Fig. 5.2. The density at the center, O, of the control volume is assumed to be ρ and the velocity there is assumed to be $\vec{V} = \hat{e}_r V_r + \hat{e}_\theta V_\theta + \hat{k} V_z$, where $\hat{e}_r$, $\hat{e}_\theta$, and $\hat{k}$ are unit vectors in the r, θ, and z directions, respectively, and V_r, V_θ, and V_z are the velocity components in the r, θ, and z directions, respectively. To evaluate $\int_{CS} \rho\vec{V}\cdot d\vec{A}$, we must consider the mass flux through each of the six faces of the control surface. The properties at each of the six faces of the control surface are obtained from a Taylor series expansion about point O. The details of the mass flux evaluation are shown in Table 5.2. Velocity components V_r, V_θ, and V_z are all assumed to be in the positive coordinate directions and we have again used the convention that the area normal is positive outwards on each face, and higher-order terms have been neglected.

We see that the net rate of mass flux out through the control surface (the term $\int_{CS} \rho\vec{V}\cdot d\vec{A}$ in Eq. 4.12) is given by

$$\left[\rho V_r + r\frac{\partial \rho V_r}{\partial r} + \frac{\partial \rho V_\theta}{\partial \theta} + r\frac{\partial \rho V_z}{\partial z}\right] dr\,d\theta\,dz$$



(a) Isometric view (b) Projection on $r\theta$ plane

Fig. 5.2 Differential control volume in cylindrical coordinates.

The mass inside the control volume at any instant is the product of the mass per unit volume, ρ, and the volume, $r\,d\theta\,dr\,dz$. Thus the rate of change of mass inside the control volume (the term $\partial/\partial t \int_{\mathrm{CV}} \rho\, d\forall$ in Eq. 4.12) is given by

$$\frac{\partial \rho}{\partial t} r\, d\theta\, dr\, dz$$

In cylindrical coordinates the differential equation for conservation of mass is then

$$\rho V_r + r\frac{\partial \rho V_r}{\partial r} + \frac{\partial \rho V_\theta}{\partial \theta} + r\frac{\partial \rho V_z}{\partial z} + r\frac{\partial \rho}{\partial t} = 0$$

or

$$\frac{\partial (r\rho V_r)}{\partial r} + \frac{\partial \rho V_\theta}{\partial \theta} + r\frac{\partial \rho V_z}{\partial z} + r\frac{\partial \rho}{\partial t} = 0$$

Dividing by r gives

$$\frac{1}{r}\frac{\partial (r\rho V_r)}{\partial r} + \frac{1}{r}\frac{\partial (\rho V_\theta)}{\partial \theta} + \frac{\partial (\rho V_z)}{\partial z} + \frac{\partial \rho}{\partial t} = 0 \tag{5.2a}$$

In cylindrical coordinates the vector operator ∇ is given by

$$\nabla = \hat{e}_r \frac{\partial}{\partial r} + \hat{e}_\theta \frac{1}{r}\frac{\partial}{\partial \theta} + \hat{k}\frac{\partial}{\partial z} \tag{3.19}$$

Equation 5.2a also may be written[1] in vector notation as

$$\nabla \cdot \rho\vec{V} + \frac{\partial \rho}{\partial t} = 0 \tag{5.1b}$$

For an *incompressible* fluid, $\rho =$ constant, and Eq. 5.2a reduces to

$$\frac{1}{r}\frac{\partial (rV_r)}{\partial r} + \frac{1}{r}\frac{\partial V_\theta}{\partial \theta} + \frac{\partial V_z}{\partial z} = \nabla \cdot \vec{V} = 0 \tag{5.2b}$$

[1]To evaluate $\nabla \cdot \rho\vec{V}$ in cylindrical coordinates, we must remember that

$$\frac{\partial \hat{e}_r}{\partial \theta} = \hat{e}_\theta \quad \text{and} \quad \frac{\partial \hat{e}_\theta}{\partial \theta} = -\hat{e}_r$$

Table 5.2

Mass Flux Through the Control Surface of a Cylindrical Differential Control Volume

Surface	Evaluation of $\int \rho \vec{V} \cdot d\vec{A}$
Inside $(-r)$	$= -\left[\rho - \left(\dfrac{\partial \rho}{\partial r}\right)\dfrac{dr}{2}\right]\left[V_r - \left(\dfrac{\partial V_r}{\partial r}\right)\dfrac{dr}{2}\right]\left(r - \dfrac{dr}{2}\right)d\theta\, dz = -\rho V_r\, r d\theta\, dz + \rho V_r \dfrac{dr}{2} d\theta\, dz + \rho\left(\dfrac{\partial V_r}{\partial r}\right) r \dfrac{dr}{2} d\theta\, dz + V_r\left(\dfrac{\partial \rho}{\partial r}\right) r \dfrac{dr}{2} d\theta\, dz$
Outside $(+r)$	$= \left[\rho + \left(\dfrac{\partial \rho}{\partial r}\right)\dfrac{dr}{2}\right]\left[V_r + \left(\dfrac{\partial V_r}{\partial r}\right)\dfrac{dr}{2}\right]\left(r + \dfrac{dr}{2}\right)d\theta\, dz = \rho V_r\, r d\theta\, dz + \rho V_r \dfrac{dr}{2} d\theta\, dz + \rho\left(\dfrac{\partial V_r}{\partial r}\right) r \dfrac{dr}{2} d\theta\, dz + V_r\left(\dfrac{\partial \rho}{\partial r}\right) r \dfrac{dr}{2} d\theta\, dz$
Front $(-\theta)$	$= -\left[\rho - \left(\dfrac{\partial \rho}{\partial \theta}\right)\dfrac{d\theta}{2}\right]\left[V_\theta - \left(\dfrac{\partial V_\theta}{\partial \theta}\right)\dfrac{d\theta}{2}\right]dr\, dz = -\rho V_\theta\, dr\, dz + \rho\left(\dfrac{\partial V_\theta}{\partial \theta}\right)\dfrac{d\theta}{2} dr\, dz + V_\theta\left(\dfrac{\partial \rho}{\partial \theta}\right)\dfrac{d\theta}{2} dr\, dz$
Back $(+\theta)$	$= \left[\rho + \left(\dfrac{\partial \rho}{\partial \theta}\right)\dfrac{d\theta}{2}\right]\left[V_\theta + \left(\dfrac{\partial V_\theta}{\partial \theta}\right)\dfrac{d\theta}{2}\right]dr\, dz = \rho V_\theta\, dr\, dz + \rho\left(\dfrac{\partial V_\theta}{\partial \theta}\right)\dfrac{d\theta}{2} dr\, dz + V_\theta\left(\dfrac{\partial \rho}{\partial \theta}\right)\dfrac{d\theta}{2} dr\, dz$
Bottom $(-z)$	$= -\left[\rho - \left(\dfrac{\partial \rho}{\partial z}\right)\dfrac{dz}{2}\right]\left[V_z - \left(\dfrac{\partial V_z}{\partial z}\right)\dfrac{dz}{2}\right]r d\theta\, dr = -\rho V_z r d\theta\, dr + \rho\left(\dfrac{\partial V_z}{\partial z}\right)\dfrac{dz}{2} r d\theta\, dr + V_z\left(\dfrac{\partial \rho}{\partial z}\right)\dfrac{dz}{2} r d\theta\, dr$
Top $(+z)$	$= \left[\rho + \left(\dfrac{\partial \rho}{\partial z}\right)\dfrac{dz}{2}\right]\left[V_z + \left(\dfrac{\partial V_z}{\partial z}\right)\dfrac{dz}{2}\right]r d\theta\, dr = \rho V_z r d\theta\, dr + \rho\left(\dfrac{\partial V_z}{\partial z}\right)\dfrac{dz}{2} r d\theta\, dr + V_z\left(\dfrac{\partial \rho}{\partial z}\right)\dfrac{dz}{2} r d\theta\, dr$

Adding the results for all six faces,

$$\int_{CS} \rho \vec{V} \cdot d\vec{A} = \left[\rho V_r + r\left\{\rho\left(\frac{\partial V_r}{\partial r}\right) + V_r\left(\frac{\partial \rho}{\partial r}\right)\right\} + \left\{\rho\left(\frac{\partial V_\theta}{\partial \theta}\right) + V_\theta\left(\frac{\partial \rho}{\partial \theta}\right)\right\} + r\left\{\rho\left(\frac{\partial V_z}{\partial z}\right) + V_z\left(\frac{\partial \rho}{\partial z}\right)\right\}\right] dr\, d\theta\, dz$$

or

$$\int_{CS} \rho \vec{V} \cdot d\vec{A} = \left[\rho V_r + r\frac{\partial \rho V_r}{\partial r} + \frac{\partial \rho V_\theta}{\partial \theta} + r\frac{\partial \rho V_z}{\partial z}\right] dr\, d\theta\, dz$$

Thus the velocity field, $\vec{V}(x, y, z, t)$, for incompressible flow must satisfy $\nabla \cdot \vec{V} = 0$. For *steady* flow, Eq. 5.2a reduces to

$$\frac{1}{r}\frac{\partial(r\rho V_r)}{\partial r} + \frac{1}{r}\frac{\partial(\rho V_\theta)}{\partial \theta} + \frac{\partial(\rho V_z)}{\partial z} = \nabla \cdot \rho\vec{V} = 0 \tag{5.2c}$$

(and remember once again that the del operator ∇ acts on ρ *and* $\vec{V}$).

When written in vector form, the differential continuity equation (the mathematical statement of conservation of mass), Eq. 5.1b, may be applied in any coordinate system. We simply substitute the appropriate expression for the vector operator ∇. In retrospect, this result is not surprising since mass must be conserved regardless of our choice of coordinate system.

Example 5.3 DIFFERENTIAL CONTINUITY EQUATION IN CYLINDRICAL COORDINATES

Consider a one-dimensional radial flow in the $r\theta$ plane, given by $V_r = f(r)$ and $V_\theta = 0$. Determine the conditions on $f(r)$ required for the flow to be incompressible.

Given: One-dimensional radial flow in the $r\theta$ plane: $V_r = f(r)$ and $V_\theta = 0$.

Find: Requirements on $f(r)$ for incompressible flow.

Solution:

Governing equation: $\nabla \cdot \rho\vec{V} + \frac{\partial \rho}{\partial t} = 0$

For incompressible flow in cylindrical coordinates this reduces to Eq. 5.2b,

$$\frac{1}{r}\frac{\partial}{\partial r}(rV_r) + \frac{1}{r}\frac{\partial}{\partial \theta}V_\theta + \frac{\partial V_z}{\partial z} = 0$$

For the given velocity field, $\vec{V} = \vec{V}(r)$. $V_\theta = 0$ and partial derivatives with respect to z are zero, so

$$\frac{1}{r}\frac{\partial}{\partial r}(rV_r) = 0$$

Integrating with respect to r gives

$$rV_r = \text{constant}$$

Thus the continuity equation shows that the radial velocity must be $V_r = f(r) = C/r$ for one-dimensional radial flow of an incompressible fluid. This is not a surprising result: As the fluid moves outwards from the center, the volume flow rate (per unit depth in the z direction) $Q = 2\pi rV$ at any radius r is constant.

*5.2 Stream Function for Two-Dimensional Incompressible Flow

We already briefly discussed streamlines in Chapter 2, where we stated that they were lines tangent to the velocity vectors in a flow at an instant

*This section may be omitted without loss of continuity in the text material.

$$\left.\frac{dy}{dx}\right|_{\text{streamline}} = \frac{v}{u} \tag{2.8}$$

We can now develop a more formal definition of streamlines by introducing the *stream function*, ψ. This will allow us to represent two entities—the velocity components $u(x, y, t)$ and $v(x, y, t)$ of a two-dimensional incompressible flow—with a single function $\psi(x, y, t)$.

There are various ways to define the stream function. We start with the two-dimensional version of the continuity equation for incompressible flow (Eq. 5.1c):

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{5.3}$$

We use what looks at first like a purely mathematical exercise (we will see a physical basis for it later) and define the stream function by

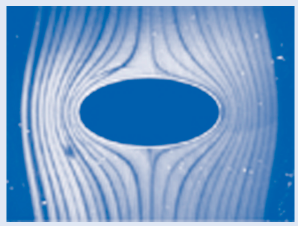
$$u \equiv \frac{\partial \psi}{\partial y} \quad \text{and} \quad v \equiv -\frac{\partial \psi}{\partial x} \tag{5.4}$$

so that Eq. 5.3 is *automatically* satisfied for *any* $\psi(x, y, t)$ we choose! To see this, use Eq. 5.4 in Eq. 5.3:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = \frac{\partial^2 \psi}{\partial x \partial y} - \frac{\partial^2 \psi}{\partial y \partial x} = 0$$

VIDEO

An Example of Streamlines/Streaklines.



Using Eq. 2.8, we can obtain an equation valid only *along* a streamline

$$udy - vdx = 0$$

or, using the definition of our stream function,

$$\frac{\partial \psi}{\partial x} dx + \frac{\partial \psi}{\partial y} dy = 0 \tag{5.5}$$

On the other hand, from a strictly mathematical point of view, at any instant in time t the variation in a function $\psi(x, y, t)$ in space (x, y) is given by

$$d\psi = \frac{\partial \psi}{\partial x} dx + \frac{\partial \psi}{\partial y} dy \tag{5.6}$$

Comparing Eqs. 5.5 and 5.6, we see that along an instantaneous streamline, $d\psi = 0$; in other words, *ψ is a constant along a streamline*. Hence we can specify individual streamlines by their stream function values: $\psi = 0, 1, 2$, etc. What is the significance of the ψ values? The answer is that they can be used to obtain the volume flow rate between any two streamlines. Consider the streamlines shown in Fig. 5.3. We can compute the volume flow rate between streamlines ψ_1 and ψ_2 by using line AB, BC, DE, or EF (recall that there is no flow *across* a streamline).

Let us compute the flow rate by using line AB, and also by using line BC—they should be the same!

For a unit depth (dimension perpendicular to the xy plane), the flow rate across AB is

$$Q = \int_{y_1}^{y_2} u\, dy = \int_{y_1}^{y_2} \frac{\partial \psi}{\partial y} dy$$

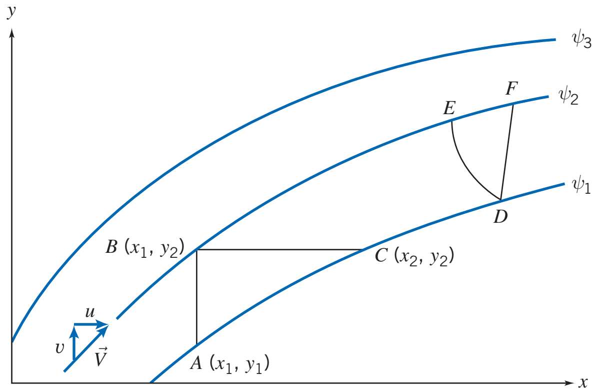


Fig. 5.3 Instantaneous streamlines in a two-dimensional flow.

But along AB, x = constant, and (from Eq. 5.6) $d\psi = \partial\psi/\partial y \; dy$. Therefore,

$$Q = \int_{y_1}^{y_2} \frac{\partial \psi}{\partial y} dy = \int_{\psi_1}^{\psi_2} d\psi = \psi_2 - \psi_1$$

For a unit depth, the flow rate across BC is

$$Q = \int_{x_1}^{x_2} v dx = -\int_{x_1}^{x_2} \frac{\partial \psi}{\partial x} dx$$

Along BC, y = constant, and (from Eq. 5.6) $d\psi = \partial\psi/\partial x \; dx$. Therefore,

$$Q = -\int_{x_1}^{x_2} \frac{\partial \psi}{\partial x} dx = -\int_{\psi_2}^{\psi_1} d\psi = \psi_2 - \psi_1$$

Hence, whether we use line AB or line BC (or for that matter lines DE or DF), we find that *the volume flow rate (per unit depth) between two streamlines is given by the difference between the two stream function values*.[2] (The derivations for lines AB and BC are the justification for using the stream function definition of Eq. 5.4.) If the streamline through the origin is designated $\psi = 0$, then the ψ value for any other streamline represents the flow between the origin and that streamline. [We are free to select any streamline as the zero streamline because the stream function is defined as a differential (Eq. 5.3); also, the flow rate will always be given by a *difference* of ψ values.] Note that because the volume flow between any two streamlines is constant, *the velocity will be relatively high wherever the streamlines are close together, and relatively low wherever the streamlines are far apart*—a very useful concept for "eyeballing" velocity fields to see where we have regions of high or low velocity.

For a two-dimensional, incompressible flow in the $r\theta$ plane, conservation of mass, Eq. 5.2b, can be written as

$$\frac{\partial (rV_r)}{\partial r} + \frac{\partial V_\theta}{\partial \theta} = 0 \qquad (5.7)$$

[2]For two-dimensional steady compressible flow in the xy plane, the stream function, ψ, can be defined such that

$$\rho u \equiv \frac{\partial \psi}{\partial y} \quad \text{and} \quad \rho v \equiv -\frac{\partial \psi}{\partial x}$$

The difference between the constant values of ψ defining two streamlines is then the mass flow rate (per unit depth) between the two streamlines.

Using a logic similar to that used for Eq. 5.4, the stream function, $\psi(r, \theta, t)$, then is defined such that

$$V_r \equiv \frac{1}{r}\frac{\partial \psi}{\partial \theta} \quad \text{and} \quad V_\theta \equiv -\frac{\partial \psi}{\partial r} \tag{5.8}$$

With ψ defined according to Eq. 5.8, the continuity equation, Eq. 5.7, is satisfied exactly.

Example 5.4 STREAM FUNCTION FOR FLOW IN A CORNER

Given the velocity field for steady, incompressible flow in a corner (Example 2.1), $\vec{V} = Ax\hat{i} - Ay\hat{j}$, with $A = 0.3\,\mathrm{s}^{-1}$, determine the stream function that will yield this velocity field. Plot and interpret the streamline pattern in the first and second quadrants of the xy plane.

Given: Velocity field, $\vec{V} = Ax\hat{i} - Ay\hat{j}$, with $A = 0.3\ \mathrm{s}^{-1}$.

Find: Stream function ψ and plot in first and second quadrants; interpret the results.

Solution:
The flow is incompressible, so the stream function satisfies Eq. 5.4.

From Eq. 5.4, $u = \dfrac{\partial \psi}{\partial y}$ and $v = -\dfrac{\partial \psi}{\partial y}$. From the given velocity field,

$$u = Ax = \frac{\partial \psi}{\partial y}$$

Integrating with respect to y gives

$$\psi = \int \frac{\partial \psi}{\partial y} dy + f(x) = Axy + f(x) \tag{1}$$

where $f(x)$ is arbitrary. The function $f(x)$ may be evaluated using the equation for v. Thus, from Eq. 1,

$$v = -\frac{\partial \psi}{\partial x} = -Ay - \frac{df}{dx} \tag{2}$$

From the given velocity field, $v = -Ay$. Comparing this with Eq. 2 shows that $\dfrac{df}{dx} = 0$, or $f(x) = \text{constant}$. Therefore, Eq. 1 becomes

$$\psi = Axy + c \longleftarrow \psi$$

Lines of constant ψ represent streamlines in the flow field. The constant c may be chosen as any convenient value for plotting purposes. The constant is chosen as zero in order that the streamline through the origin be designated as $\psi = \psi_1 = 0$. Then the value for any other streamline represents the flow between the origin and that streamline. With $c = 0$ and $A = 0.3\ \mathrm{s}^{-1}$, then

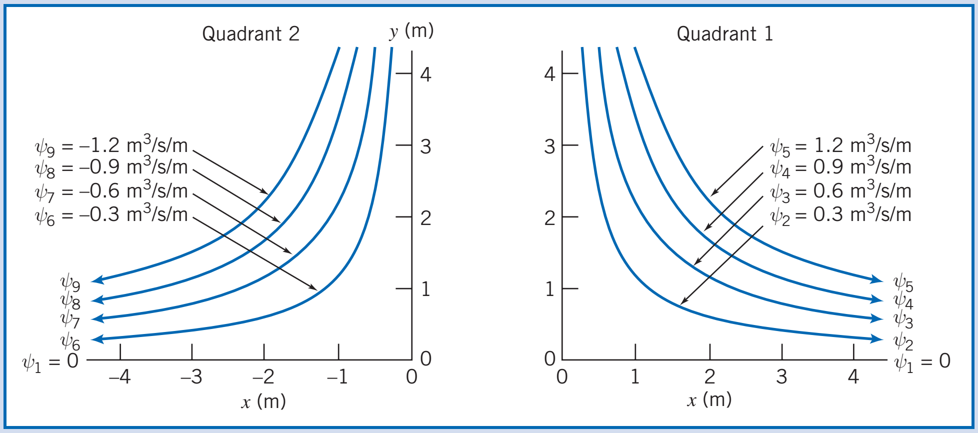
$$\psi = 0.3xy \quad (\mathrm{m}^3/\mathrm{s}/\mathrm{m})$$

{This equation of a streamline is identical to the result ($xy = \text{constant}$) obtained in Example 2.1.}

Separate plots of the streamlines in the first and second quadrants are presented below. Note that in quadrant 1, $u > 0$, so ψ values are positive. In quadrant 2, $u < 0$, so ψ values are negative.

In the first quadrant, since $u > 0$ and $v < 0$, the flow is from left to right and down. The volume flow rate between the streamline $\psi = \psi_1$ through the origin and the streamline $\psi = \psi_2$ is

$$Q_{12} = \psi_2 - \psi_1 = 0.3\,\mathrm{m}^3/\mathrm{s}/\mathrm{m}$$



In the second quadrant, since $u<0$ and $v<0$, the flow is from right to left and down. The volume flow rate between streamlines ψ_7 and ψ_9 is

$$Q_{79} = \psi_9 - \psi_7 = [-1.2 - (-0.6)]\,\mathrm{m^3/s/m} = -0.6\,\mathrm{m^3/s/m}$$

The negative sign is consistent with flow having $u<0$.

As both the streamline spacing in the graphs and the equation for $\vec{V}$ indicate, the velocity is smallest near the origin (a "corner").

There is an *Excel* workbook for this problem that can be used to generate streamlines for this and many other stream functions.

5.3 Motion of a Fluid Particle (Kinematics)

Figure 5.4 shows a typical finite fluid element, within which we have selected an infinitesimal particle of mass dm and initial volume $dx\ dy\ dz$, at time t, and as it (and the infinitesimal particle) may appear after a time interval dt. The finite element has moved and changed its shape and orientation. Note that while the finite element has quite severe distortion, the infinitesimal particle has changes in shape limited to stretching/shrinking and rotation of the element's sides—this is because we are

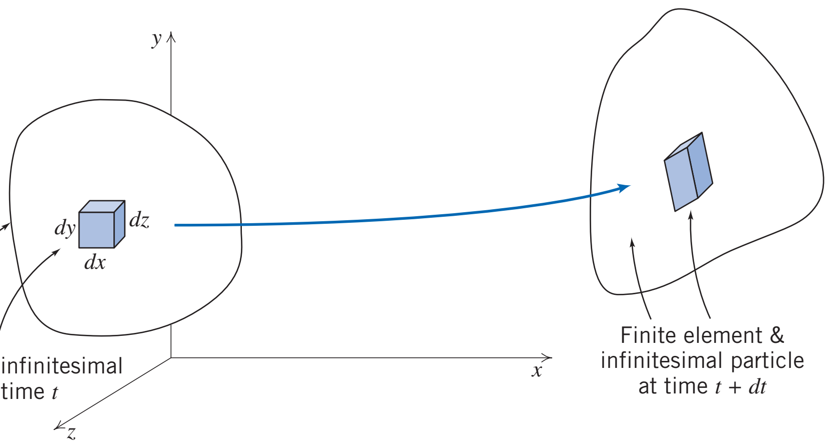


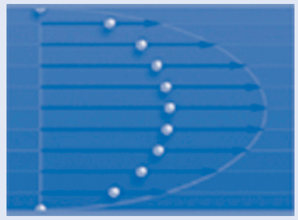
Fig. 5.4 Finite fluid element and infinitesimal particle at times t and $t + dt$.

considering both an infinitesimal time step and particle, so that the sides remain straight. We will examine the infinitesimal particle so that we will eventually obtain results applicable to a point. We can decompose this particle's motion into four components: *translation*, in which the particle moves from one point to another; *rotation* of the particle, which can occur about any or all of the x, y or z axes; *linear deformation*, in which the particle's sides stretch or contract; and *angular deformation*, in which the angles (which were initially 90° for our particle) between the sides change.

It may seem difficult by looking at Fig. 5.4 to distinguish between rotation and angular deformation of the infinitesimal fluid particle. It is important to do so, because pure rotation involves no deformation but angular deformation does and, as we learned in Chapter 2, fluid deformation generates shear stresses. Figure 5.5 shows a typical xy plane motion decomposed into the four components described above, and as we examine each of these four components in turn we will see that we *can* distinguish between rotation and angular deformation.

VIDEO

Particle Motion in a Channel.



VIDEO

Particle Motion over a Cylinder.

Fluid Translation: Acceleration of a Fluid Particle in a Velocity Field

The translation of a fluid particle is obviously connected with the velocity field $\vec{V} = \vec{V}(x, y, z, t)$ that we previously discussed in Section 2.2. We will need the acceleration of a fluid particle for use in Newton's second law. It might seem that we could simply compute this as $\vec{a} = \partial\vec{V}/\partial t$. This is incorrect, because $\vec{V}$ is a *field*, i.e., it describes the whole flow and not just the motion of an individual particle. (We can see that this way of computing is incorrect by examining Example 5.4, in which particles are clearly accelerating and decelerating so $\vec{a} \neq 0$, but $\partial\vec{V}/\partial t = 0$.)

The problem, then, is to retain the field description for fluid properties and obtain an expression for the acceleration of a fluid particle as it moves in a flow field. Stated simply, the problem is:

> Given the velocity field, $\vec{V} = \vec{V}(x, y, z, t)$, find the acceleration of a fluid particle, $\vec{a}_p$.

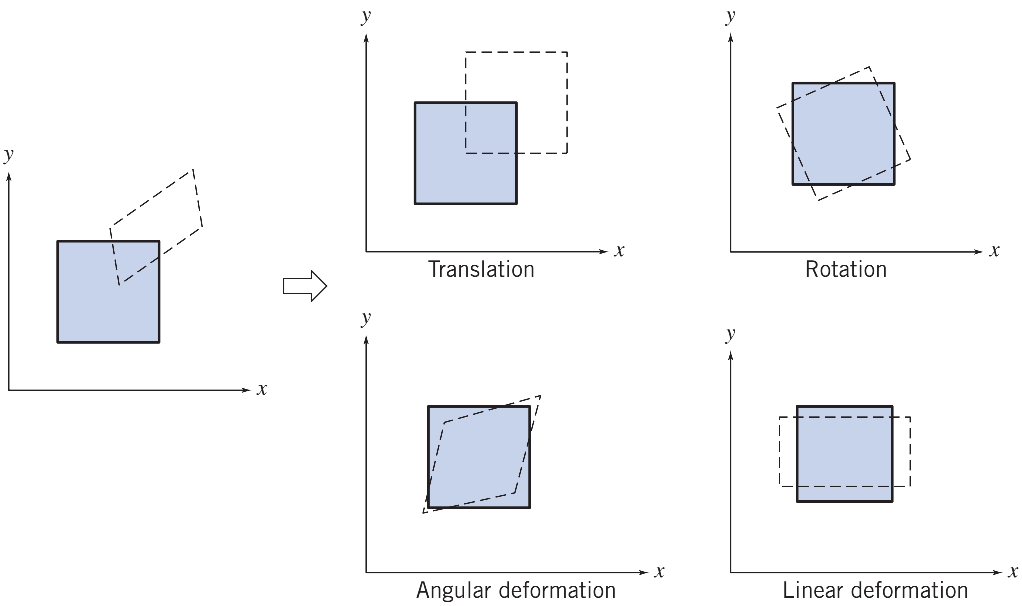


Fig. 5.5 Pictorial representation of the components of fluid motion.

Consider a particle moving in a velocity field. At time t, the particle is at the position x, y, z and has a velocity corresponding to the velocity at that point in space at time t,

$$\vec{V}_p\Big]_t = \vec{V}(x, y, z, t)$$

At $t + dt$, the particle has moved to a new position, with coordinates $x + dx$, $y + dy$, $z + dz$, and has a velocity given by

$$\vec{V}_p\Big]_{t+dt} = \vec{V}(x + dx, y + dy, z + dz, t + dt)$$

This is shown pictorially in Fig. 5.6.

The particle velocity at time t (position $\vec{r}$) is given by $\vec{V}_p = \vec{V}(x, y, z, t)$. Then $d\vec{V}_p$, the change in the velocity of the particle, in moving from location $\vec{r}$ to $\vec{r} + d\vec{r}$, in time dt, is given by the chain rule,

$$d\vec{V}_p = \frac{\partial \vec{V}}{\partial x} dx_p + \frac{\partial \vec{V}}{\partial y} dy_p + \frac{\partial \vec{V}}{\partial z} dz_p + \frac{\partial \vec{V}}{\partial t} dt$$

The total acceleration of the particle is given by

$$\vec{a}_p = \frac{d\vec{V}_p}{dt} = \frac{\partial \vec{V}}{\partial x}\frac{dx_p}{dt} + \frac{\partial \vec{V}}{\partial y}\frac{dy_p}{dt} + \frac{\partial \vec{V}}{\partial z}\frac{dz_p}{dt} + \frac{\partial \vec{V}}{\partial t}$$

Since

$$\frac{dx_p}{dt} = u, \qquad \frac{dy_p}{dt} = v, \qquad \text{and} \qquad \frac{dz_p}{dt} = w,$$

we have

$$\vec{a}_p = \frac{d\vec{V}_p}{dt} = u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + w\frac{\partial \vec{V}}{\partial z} + \frac{\partial \vec{V}}{\partial t}$$

To remind us that calculation of the acceleration of a fluid particle in a velocity field requires a special derivative, it is given the symbol $D\vec{V}/Dt$. Thus

$$\frac{D\vec{V}}{Dt} \equiv \vec{a}_p = u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + w\frac{\partial \vec{V}}{\partial z} + \frac{\partial \vec{V}}{\partial t} \tag{5.9}$$

The derivative, D/Dt, defined by Eq. 5.9, is commonly called the *substantial derivative* to remind us that it is computed for a particle of "substance." It often is called the *material derivative* or *particle derivative*.

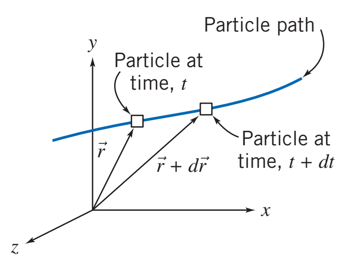


Fig. 5.6 Motion of a particle in a flow field.

The physical significance of the terms in Eq. 5.9 is

$$\underbrace{\vec{a}_p = \frac{D\vec{V}}{Dt}}_{\substack{\text{total}\\ \text{acceleration}\\ \text{of a particle}}} = \underbrace{u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + w\frac{\partial \vec{V}}{\partial z}}_{\substack{\text{convective}\\ \text{acceleration}}} + \underbrace{\frac{\partial \vec{V}}{\partial t}}_{\substack{\text{local}\\ \text{acceleration}}}$$

From Eq. 5.9 we recognize that a fluid particle moving in a flow field may undergo acceleration for either of two reasons. As an illustration, refer to Example 5.4. This is a steady flow in which particles are *convected* toward the low-velocity region (near the "corner"), and then away to a high-velocity region. If a flow field is unsteady a fluid particle will undergo an additional *local* acceleration, because the velocity field is a function of time.

The convective acceleration may be written as a single vector expression using the gradient operator ∇. Thus

$$u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + w\frac{\partial \vec{V}}{\partial z} = (\vec{V} \cdot \nabla)\vec{V}$$

(We suggest that you check this equality by expanding the right side of the equation using the familiar dot product operation.) Thus Eq. 5.9 may be written as

$$\frac{D\vec{V}}{Dt} \equiv \vec{a}_p = (\vec{V} \cdot \nabla)\vec{V} + \frac{\partial \vec{V}}{\partial t} \tag{5.10}$$

For a *two-dimensional flow*, say $\vec{V} = \vec{V}(x, y, t)$, Eq. 5.9 reduces to

$$\frac{D\vec{V}}{Dt} = u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + \frac{\partial \vec{V}}{\partial t}$$

For a *one-dimensional flow*, say $\vec{V} = \vec{V}(x, t)$, Eq. 5.9 becomes

$$\frac{D\vec{V}}{Dt} = u\frac{\partial \vec{V}}{\partial x} + \frac{\partial \vec{V}}{\partial t}$$

Finally, for a *steady flow in three dimensions*, Eq. 5.9 becomes

$$\frac{D\vec{V}}{Dt} = u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + w\frac{\partial \vec{V}}{\partial z}$$

which, as we have seen, is not necessarily zero even though the flow is steady. Thus a fluid particle may undergo a convective acceleration due to its motion, even in a steady velocity field.

Equation 5.9 is a vector equation. As with all vector equations, it may be written in scalar component equations. Relative to an xyz coordinate system, the scalar components of Eq. 5.9 are written

$$a_{x_p} = \frac{Du}{Dt} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t} \tag{5.11a}$$

$$a_{y_p} = \frac{Dv}{Dt} = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} + \frac{\partial v}{\partial t} \tag{5.11b}$$

$$a_{z_p} = \frac{Dw}{Dt} = u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} + \frac{\partial w}{\partial t} \tag{5.11c}$$

CLASSIC VIDEO

Eulerian and Lagrangian Descriptions in Fluid Mechanics.

The components of acceleration in cylindrical coordinates may be obtained from Eq. 5.10 by expressing the velocity, $\vec{V}$, in cylindrical coordinates (Section 5.1) and utilizing the appropriate expression (Eq. 3.19, on the Web) for the vector operator ∇. Thus,[3]

$$a_{r_p} = V_r \frac{\partial V_r}{\partial r} + \frac{V_\theta}{r}\frac{\partial V_r}{\partial \theta} - \frac{V_\theta^2}{r} + V_z \frac{\partial V_r}{\partial z} + \frac{\partial V_r}{\partial t} \tag{5.12a}$$

$$a_{\theta_p} = V_r \frac{\partial V_\theta}{\partial r} + \frac{V_\theta}{r}\frac{\partial V_\theta}{\partial \theta} + \frac{V_r V_\theta}{r} + V_z \frac{\partial V_\theta}{\partial z} + \frac{\partial V_\theta}{\partial t} \tag{5.12b}$$

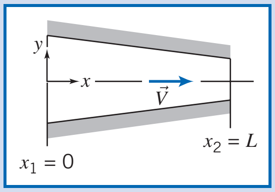
$$a_{z_p} = V_r \frac{\partial V_z}{\partial r} + \frac{V_\theta}{r}\frac{\partial V_z}{\partial \theta} + V_z \frac{\partial V_z}{\partial z} + \frac{\partial V_z}{\partial t} \tag{5.12c}$$

Equations 5.9, 5.11, and 5.12 are useful for computing the acceleration of a fluid particle anywhere in a flow from the velocity field (a function of x, y, z, and t); this is the *Eulerian* method of description, the most-used approach in fluid mechanics.

As an alternative (e.g., if we wish to track an individual particle's motion in, for example, pollution studies) we sometimes use the *Lagrangian* description of particle motion, in which the acceleration, position, and velocity of a particle are specified as a function of time only. Both descriptions are illustrated in Example 5.5.

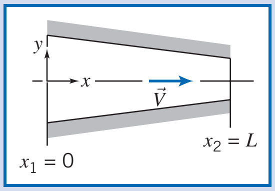
Example 5.5 PARTICLE ACCELERATION IN EULERIAN AND LAGRANGIAN DESCRIPTIONS

Consider two-dimensional, steady, incompressible flow through the plane converging channel shown. The velocity on the horizontal centerline (x axis) is given by $\vec{V} = V_1[1 + (x/L)]\hat{i}$. Find an expression for the acceleration of a particle moving along the centerline using (a) the Eulerian approach and (b) the Lagrangian approach. Evaluate the acceleration when the particle is at the beginning and at the end of the channel.



Given: Steady, two-dimensional, incompressible flow through the converging channel shown.

$$\vec{V} = V_1\left(1 + \frac{x}{L}\right)\hat{i} \quad \text{on } x \text{ axis}$$



Find: (a) The acceleration of a particle moving along the centerline using the Eulerian approach.
(b) The acceleration of a particle moving along the centerline using the Lagrangian approach.
(c) Evaluate the acceleration when the particle is at the beginning and at the end of the channel.

Solution:

(a) The Eulerian approach

The governing equation for acceleration of a fluid particle is Eq. 5.9:

$$\vec{a}_p(x, y, z, t) = \frac{D\vec{V}}{Dt} = u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + w\frac{\partial \vec{V}}{\partial z} + \frac{\partial \vec{V}}{\partial t} \tag{5.9}$$

[3]In evaluating $(\vec{V}\cdot\nabla)\vec{V}$, recall that $\hat{e}_r$ and $\hat{e}_\theta$ are functions of θ (see footnote 1 on p. 178).

In this case we are interested in the x component of acceleration (Eq. 5.11a):

$$a_{x_p}(x,y,z,t) = \frac{Du}{Dt} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t} \tag{5.11a}$$

On the x axis, $v = w = 0$ and $u = V_1\left(1+\frac{x}{L}\right)$, so for steady flow we obtain

$$a_{x_p}(x) = \frac{Du}{Dt} = u\frac{\partial u}{\partial x} = V_1\left(1+\frac{x}{L}\right)\frac{V_1}{L}$$

or

$$a_{x_p}(x) = \frac{V_1^2}{L}\left(1+\frac{x}{L}\right) \longleftarrow a_{x_p}(x)$$

This expression gives the acceleration of *any* particle that is at point x at an instant.

(b) The Lagrangian approach
In this approach we need to obtain the motion of a fluid particle as we would in particle mechanics; that is, we need the position $\vec{x}_p(t)$, and then we can obtain the velocity $\vec{V}_p(t) = d\vec{x}_p/dt$ and acceleration $\vec{a}_p(t) = d\vec{V}_p/dt$. Actually, we are considering motion along the x axis, so we want $x_p(t)$, $u_p(t) = dx_p/dt$, and $a_{x_p}(t) = du_p/dt$. We are not given $x_p(t)$, but we do have

$$u_p = \frac{dx_p}{dt} = V_1\left(1+\frac{x_p}{L}\right)$$

Separating variables, and using limits $x_p(t=0)=0$ and $x_p(t=t)=x_p$,

$$\int_0^{x_p}\frac{dx_p}{\left(1+\frac{x_p}{L}\right)} = \int_0^1 V_1\,dt \qquad \text{and} \qquad L\ln\left(1+\frac{x_p}{L}\right) = V_1 t \tag{1}$$

We can then solve for $x_p(t)$:

$$x_p(t) = L(e^{V_1 t/L} - 1)$$

The velocity and acceleration are then

$$u_p(t) = \frac{dx_p}{dt} = V_1 e^{V_1 t/L}$$

and

$$a_{x_p}(t) = \frac{du_p}{dt} = \frac{V_1^2}{L}e^{V_1 t/L} \longleftarrow \text{(2)} \quad a_{x_p}(t)$$

This expression gives the acceleration at any time t of the particle that was initially at $x=0$.

(c) We wish to evaluate the acceleration when the particle is at $x=0$ and $x=L$. For the Eulerian approach this is straightforward:

$$a_{z_p}(x=0) = \frac{V_1^2}{L}, \qquad a_{x_p}(x=L) = 2\frac{V_1^2}{L} \longleftarrow a_{x_p}$$

For the Lagrangian approach, we need to find the times at which $x=0$ and $x=L$. Using Eq. 1, these are

$$t(x_p=0) = \frac{L}{V_1} \qquad t(x_p=L) = \frac{L}{V_1}\ln(2)$$

Then, from Eq. 2,

$$a_{z_p}(t=0) = \frac{V_1^2}{L}, \qquad \text{and}$$

$$a_{x_p}\left(t = \frac{L}{V_1}\ln(2)\right) = \frac{V_1^2}{L}e^{\ln(2)} = 2\frac{V_1^2}{L} \longleftarrow a_{x_p}$$

Note that both approaches yield the same results for particle acceleration, as they should.

This problem illustrates use of the Eulerian and Lagrangian descriptions of the motion of a fluid particle.

Fluid Rotation

A fluid particle moving in a general three-dimensional flow field may rotate about all three coordinate axes. Thus particle rotation is a vector quantity and, in general,

$$\vec{\omega} = \hat{i}\omega_x + \hat{j}\omega_y + \hat{k}\omega_z$$

where ω_x is the rotation about the x axis, ω_y is the rotation about the y axis, and ω_z is the rotation about the z axis. The positive sense of rotation is given by the right-hand rule.

We now see how we can extract the rotation component of the particle motion. Consider the xy plane view of the particle at time t. The left and lower sides of the particle are given by the two perpendicular line segments oa and ob of lengths Δx and Δy, respectively, shown in Fig. 5.7*a*. In general, after an interval Δt the particle will have translated to some new position, and also have rotated and deformed. A possible instantaneous orientation of the lines at time $t+\Delta t$ is shown in Fig. 5.7*b*.

We need to be careful here with our signs for angles. Following the right-hand rule, *counterclockwise rotation is positive*, and we have shown side oa rotating counterclockwise through angle $\Delta\alpha$, but be aware that we have shown edge ob rotating at a *clockwise* angle $\Delta\beta$. Both angles are obviously arbitrary, but it will help visualize the discussion if we assign values to these angles, e.g., let $\Delta\alpha = 6°$ and $\Delta\beta = 4°$.

How do we extract from $\Delta\alpha$ and $\Delta\beta$ a measure of the particle's rotation? The answer is that we take an average of the rotations $\Delta\alpha$ and $\Delta\beta$, so that the particle's rigid body counterclockwise rotation is $\frac{1}{2}(\Delta\alpha - \Delta\beta)$, as shown in Fig. 5.7*c*. The minus sign is needed because the *counterclockwise* rotation of ob is $-\Delta\beta$. Using the assigned values, the rotation of the particle is then $\frac{1}{2}(6° - 4°) = 1°$. (Given the two rotations, taking the average is the only way we can measure the particle's rotation, because any other approach would favor one side's rotation over the other, which doesn't make sense.)

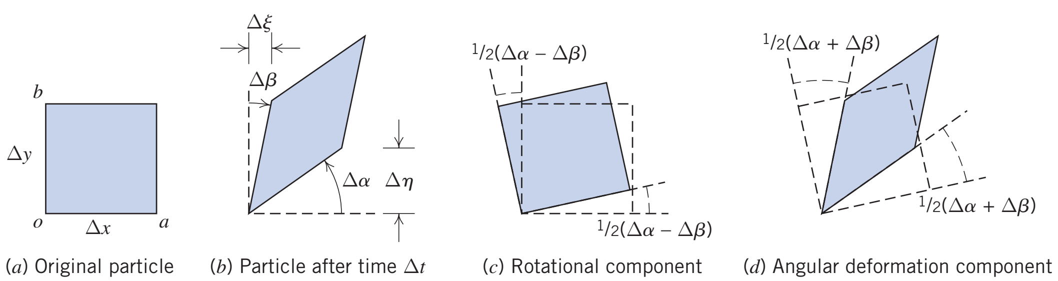


Fig. 5.7 Rotation and angular deformation of perpendicular line segments in a two-dimensional flow.

Now we can determine from $\Delta\alpha$ and $\Delta\beta$ a measure of the particle's angular deformation, as shown in Fig. 5.7*d*. To obtain the deformation of side *oa* in Fig. 5.7*d*, we use Fig. 5.7*b* and 5.7*c*: If we subtract the particle rotation $\frac{1}{2}(\Delta\alpha - \Delta\beta)$, in Fig. 5.7*c*, from the actual rotation of *oa*, $\Delta\alpha$, in Fig. 5.7*b*, what remains must be pure deformation $[\Delta\alpha - \frac{1}{2}(\Delta\alpha - \Delta\beta) = \frac{1}{2}(\Delta\alpha + \Delta\beta)$, in Fig. 5.7*d*]. Using the assigned values, the deformation of side *oa* is $6° - \frac{1}{2}(6° - 4°) = 5°$. By a similar process, for side *ob* we end with $\Delta\beta - \frac{1}{2}(\Delta\alpha - \Delta\beta) = -\frac{1}{2}(\Delta\alpha + \Delta\beta)$, or a *clockwise* deformation $\frac{1}{2}(\Delta\alpha + \Delta\beta)$, as shown in Fig. 5.7*d*. The total deformation of the particle is the sum of the deformations of the sides, or $(\Delta\alpha + \Delta\beta)$ (with our example values, 10°). We verify that this leaves us with the correct value for the particle's deformation: Recall that in Section 2.4 we saw that deformation is measured by the change in a 90° angle. In Fig. 5.7*a* we see this is angle *aob*, and in Fig. 5.7*d* we see the total change of this angle is indeed $\frac{1}{2}(\Delta\alpha + \Delta\beta) + \frac{1}{2}(\Delta\alpha + \Delta\beta) = (\Delta\alpha + \Delta\beta)$.

We need to convert these angular measures to quantities obtainable from the flow field. To do this, we recognize that (for small angles) $\Delta\alpha = \Delta\eta/\Delta x$, and $\Delta\beta = \Delta\xi/\Delta y$. But $\Delta\xi$ arises because, if in interval Δt point *o* moves horizontally distance $u\Delta t$, then point *b* will have moved distance $(u + [\partial u/\partial y]\Delta y)\Delta t$ (using a Taylor series expansion). Likewise, $\Delta\eta$ arises because, if in interval Δt point *o* moves vertically distance $v\Delta t$, then point *a* will have moved distance $(v + [\partial v/\partial x]\Delta x)\Delta t$. Hence,

$$\Delta\xi = \left(u + \frac{\partial u}{\partial y}\Delta y\right)\Delta t - u\Delta t = \frac{\partial u}{\partial y}\Delta y\Delta t$$

and

$$\Delta\eta = \left(v + \frac{\partial v}{\partial x}\Delta x\right)\Delta t - v\Delta t = \frac{\partial v}{\partial x}\Delta x\Delta t$$

We can now compute the angular velocity of the particle about the z axis, ω_z, by combining all these results:

$$\omega_z = \lim_{\Delta t\to 0}\frac{\frac{1}{2}(\Delta\alpha - \Delta\beta)}{\Delta t} = \lim_{\Delta t\to 0}\frac{\frac{1}{2}\left(\frac{\Delta\eta}{\Delta x} - \frac{\Delta\xi}{\Delta y}\right)}{\Delta t} = \lim_{\Delta t\to 0}\frac{\frac{1}{2}\left(\frac{\partial v}{\partial x}\frac{\Delta x}{\Delta x}\Delta t - \frac{\partial u}{\partial y}\frac{\Delta y}{\Delta y}\Delta t\right)}{\Delta t}$$

$$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)$$

By considering the rotation of pairs of perpendicular line segments in the yz and xz planes, one can show similarly that

$$\omega_x = \frac{1}{2}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right) \quad \text{and} \quad \omega_y = \frac{1}{2}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)$$

Then $\vec{\omega} = \hat{i}\omega_x + \hat{j}\omega_y + \hat{k}\omega_z$ becomes

$$\vec{\omega} = \frac{1}{2}\left[\hat{i}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right) + \hat{j}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right) + \hat{k}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\right] \tag{5.13}$$

We recognize the term in the square brackets as

$$\text{curl}\,\vec{V} = \nabla \times \vec{V}$$

Then, in vector notation, we can write

$$\vec{\omega} = \frac{1}{2}\nabla \times \vec{V} \qquad (5.14)$$

It is worth noting here that we should not confuse rotation of a fluid particle with flow consisting of circular streamlines, or *vortex* flow. As we will see in Example 5.6, in such a flow the particles *could* rotate as they move in a circular motion, but they do not have to!

When might we expect to have a flow in which the particles rotate as they move ($\vec{\omega} \neq 0$)? One possibility is that we start out with a flow in which (for whatever reason) the particles already have rotation. On the other hand, if we assumed the particles are not initially rotating, particles will only begin to rotate if they experience a torque caused by surface shear stresses; the particle body forces and normal (pressure) forces may accelerate and deform the particle, but cannot generate a torque. We can conclude that rotation of fluid particles will *always* occur for flows in which we have shear stresses. We have already learned in Chapter 2 that shear stresses are present whenever we have a viscous fluid that is experiencing angular deformation (shearing). Hence we conclude that rotation of fluid particles only occurs in viscous flows[4] (unless the particles are initially rotating, as in Example 3.10).

CLASSIC VIDEO

Vorticity.

Flows for which no particle rotation occurs are called *irrotational* flows. Although no real flow is truly irrotational (all fluids have viscosity), it turns out that many flows can be successfully studied by assuming they are inviscid and irrotational, because viscous effects are often negligible. As we discussed in Chapter 1, and will again in Chapter 6, much of aerodynamics theory assumes inviscid flow. We just need to be aware that in any flow there will always be regions (e.g., the boundary layer for flow over a wing) in which viscous effects cannot be ignored.

The factor of $\frac{1}{2}$ can be eliminated from Eq. 5.14 by defining the *vorticity*, $\vec{\zeta}$, to be twice the rotation,

$$\vec{\zeta} \equiv 2\vec{\omega} = \nabla \times \vec{V} \qquad (5.15)$$

The vorticity is a measure of the rotation of a fluid element as it moves in the flow field. In cylindrical coordinates the vorticity is[5]

$$\nabla \times \vec{V} = \hat{e}_r\left(\frac{1}{r}\frac{\partial V_z}{\partial \theta} - \frac{\partial V_\theta}{\partial z}\right) + \hat{e}_\theta\left(\frac{\partial V_r}{\partial z} - \frac{\partial V_z}{\partial r}\right) + \hat{k}\left(\frac{1}{r}\frac{\partial r V_\theta}{\partial r} - \frac{1}{r}\frac{\partial V_r}{\partial \theta}\right) \qquad (5.16)$$

The *circulation*, Γ (which we will revisit in Example 6.12), is defined as the line integral of the tangential velocity component about any closed curve fixed in the flow,

$$\Gamma = \oint_c \vec{V} \cdot d\vec{s} \qquad (5.17)$$

where $d\vec{s}$ is an elemental vector tangent to the curve and having length ds of the element of arc; a positive sense corresponds to a counterclockwise path of integration around the curve. We can develop a relationship between circulation and vorticity by considering the rectangular circuit shown in Fig. 5.8, where the velocity components at o are assumed to be (u, v), and the velocities along segments bc and ac can be derived using Taylor series approximations.

[4]A rigorous proof using the complete equations of motion for a fluid particle is given in Li and Lam, pp. 142–145.

[5]In carrying out the curl operation, recall that $\hat{e}_r$ and $\hat{e}_\theta$ are functions of θ (see footnote 1 on p. 178).

For the closed curve *oacb*,

$$\Delta\Gamma = u\Delta x + \left(v + \frac{\partial v}{\partial x}\Delta x\right)\Delta y - \left(u + \frac{\partial u}{\partial y}\Delta y\right)\Delta x - v\,\Delta y$$

$$\Delta\Gamma = \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\Delta x\Delta y$$

$$\Delta\Gamma = 2\omega_z \Delta x\Delta y$$

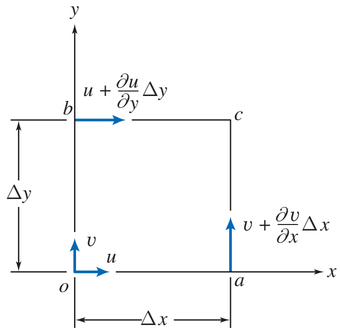


Fig. 5.8 Velocity components on the boundaries of a fluid element.

Then,

$$\Gamma = \oint_c \vec{V}\cdot d\vec{s} = \int_A 2\omega_z\, dA = \int_A (\nabla \times \vec{V})_z\, dA \tag{5.18}$$

Equation 5.18 is a statement of the Stokes Theorem in two dimensions. Thus the circulation around a closed contour is equal to the total vorticity enclosed within it.

Example 5.6 FREE AND FORCED VORTEX FLOWS

Consider flow fields with purely tangential motion (circular streamlines): $V_r = 0$ and $V_\theta = f(r)$. Evaluate the rotation, vorticity, and circulation for rigid-body rotation, a *forced vortex*. Show that it is possible to choose $f(r)$ so that flow is irrotational, i.e., to produce a *free vortex*.

Given: Flow fields with tangential motion, $V_r = 0$ and $V_\theta = f(r)$.

Find: (a) Rotation, vorticity, and circulation for rigid-body motion (a *forced vortex*).
(b) $V_\theta = f(r)$ for irrotational motion (a *free vortex*).

Solution:

Governing equation: $\vec{\zeta} = 2\vec{\omega} = \nabla \times \vec{V}$ (5.15)

For motion in the $r\theta$ plane, the only components of rotation and vorticity are in the z direction,

$$\zeta_z = 2\omega_z = \frac{1}{r}\frac{\partial rV_\theta}{\partial r} - \frac{1}{r}\frac{\partial V_r}{\partial \theta}$$

Because $V_r = 0$ everywhere in these fields, this reduces to $\zeta_z = 2\omega_z = \dfrac{1}{r}\dfrac{\partial rV_\theta}{\partial r}$.

(a) For rigid-body rotation, $V_\theta = \omega r$.

Then $\omega_z = \dfrac{1}{2}\dfrac{1}{r}\dfrac{\partial rV_\theta}{\partial r} = \dfrac{1}{2}\dfrac{1}{r}\dfrac{\partial}{\partial r}\omega r^2 = \dfrac{1}{2r}(2\omega r) = \omega$ and $\zeta_z = 2\omega$.

The circulation is $\Gamma = \oint_c \vec{V}\cdot d\vec{s} = \int_A 2\omega_z dA.$ (5.18)

Since $\omega_z = \omega =$ constant, the circulation about any closed contour is given by $\Gamma = 2\omega A$, where A is the area enclosed by the contour. Thus for rigid-body motion (a forced vortex), the rotation and vorticity are constants; the circulation depends on the area enclosed by the contour.

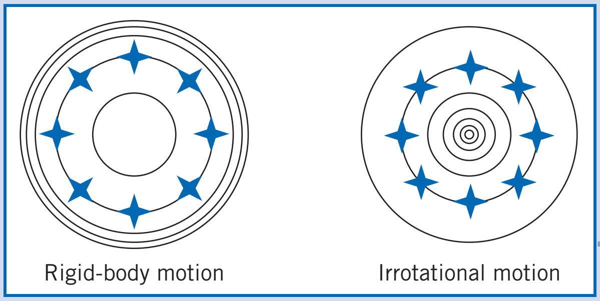
(b) For irrotational flow, $\omega_z = \dfrac{1}{r}\dfrac{\partial}{\partial r}rV_\theta = 0$. Integrating, we find

$$rV_\theta = \text{constant} \quad \text{or} \quad V_\theta = f(r) = \frac{C}{r}$$

For this flow, the origin is a singular point where $V_\theta \to \infty$. The circulation for any contour enclosing the origin is

$$\Gamma = \oint_c \vec{V} \cdot d\vec{s} = \int_0^{2\pi} \frac{C}{r} r\, d\theta = 2\pi C$$

It turns out that the circulation around any contour *not* enclosing the singular point at the origin is zero. Streamlines for the two vortex flows are shown below, along with the location and orientation at different instants of a cross marked in the fluid that was initially at the 12 o'clock position. For the rigid-body motion (which occurs, for example, at the eye of a tornado, creating the "dead" region at the very center), the cross rotates as it moves in a circular motion; also, the streamlines are closer together as we move away from the origin. For the irrotational motion (which occurs, for example, outside the eye of a tornado—in such a large region viscous effects are negligible), the cross does not rotate as it moves in a circular motion; also, the streamlines are farther apart as we move away from the origin.



Fluid Deformation

a. Angular Deformation

As we discussed earlier (and as shown in Fig. 5.7*d*), the *angular deformation* of a particle is given by the sum of the two angular deformations, or in other words by $(\Delta\alpha + \Delta\beta)$.

We also recall that $\Delta\alpha = \Delta\eta/\Delta x$, $\Delta\beta = \Delta\xi/\Delta y$, and $\Delta\xi$ and $\Delta\eta$ are given by

$$\Delta\xi = \left(u + \frac{\partial u}{\partial y}\Delta y\right)\Delta t - u\Delta t = \frac{\partial u}{\partial y}\Delta y \Delta t$$

and

$$\Delta\eta = \left(v + \frac{\partial v}{\partial x}\Delta x\right)\Delta t - v\Delta t = \frac{\partial v}{\partial x}\Delta x \Delta t$$

We can now compute the rate of angular deformation of the particle in the xy plane by combining these results,

$$\begin{array}{l}\text{Rate of angular}\\ \text{deformation}\\ \text{in } xy \text{ plane}\end{array} = \lim_{\Delta t \to 0} \frac{(\Delta\alpha + \Delta\beta)}{\Delta t} = \lim_{\Delta t \to 0} \frac{\left(\dfrac{\Delta\eta}{\Delta x} + \dfrac{\Delta\xi}{\Delta y}\right)}{\Delta t}$$

$$\begin{array}{l}\text{Rate of angular}\\ \text{deformation}\\ \text{in } xy \text{ plane}\end{array} = \lim_{\Delta t \to 0} \frac{\left(\dfrac{\partial v}{\partial x}\dfrac{\Delta x}{\Delta x}\Delta t + \dfrac{\partial u}{\partial y}\dfrac{\Delta y}{\Delta y}\Delta t\right)}{\Delta t} = \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right) \qquad (5.19a)$$

Similar expressions can be written for the rate of angular deformation of the particle in the yz and zx planes,

$$\text{Rate of angular deformation in } yz \text{ plane} = \left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right) \tag{5.19b}$$

$$\text{Rate of angular deformation in } zx \text{ plane} = \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right) \tag{5.19c}$$

CLASSIC VIDEO

Deformation of Continuous Media.

We saw in Chapter 2 that for one-dimensional laminar Newtonian flow the shear stress is given by the rate of deformation (du/dy) of the fluid particle,

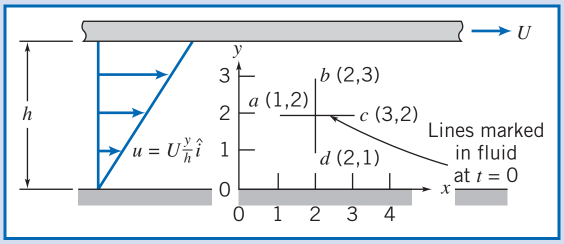
$$\tau_{yx} = \mu\frac{du}{dy} \tag{2.15}$$

We will see shortly that we can generalize Eq. 2.15 to the case of three-dimensional laminar flow; this will lead to expressions for three-dimensional shear stresses involving the three rates of angular deformation given above. (Eq. 2.15 is a special case of Eq. 5.19a.)

Calculation of angular deformation is illustrated for a simple flow field in Example 5.7.

Example 5.7 ROTATION IN VISCOMETRIC FLOW

A viscometric flow in the narrow gap between large parallel plates is shown. The velocity field in the narrow gap is given by $\vec{V} = U(y/h)\hat{i}$, where $U = 4$ mm/s and $h = 4$ mm. At $t = 0$ line segments ac and bd are marked in the fluid to form a cross as shown. Evaluate the positions of the marked points at $t = 1.5$ s and sketch for comparison. Calculate the rate of angular deformation and the rate of rotation of a fluid particle in this velocity field. Comment on your results.



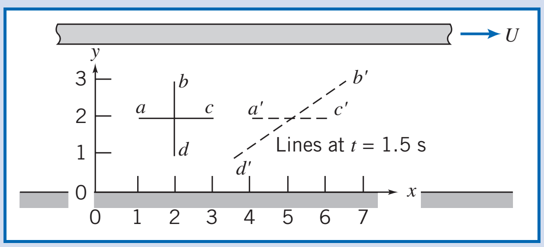
Given: Velocity field, $\vec{V} = U(y/h)\hat{i}$; $U = 4$ mm/s, and $h = 4$ mm. Fluid particles marked at $t = 0$ to form cross as shown.

Find:
(a) Positions of points a', b', c', and d' at $t = 1.5$ s; plot.
(b) Rate of angular deformation.
(c) Rate of rotation of a fluid particle.
(d) Significance of these results.

Solution: For the given flow field $v = 0$, so there is no vertical motion. The velocity of each point stays constant, so $\Delta x = u\Delta t$ for each point. At point b, $u = 3$ mm/s, so

$$\Delta x_b = 3\,\frac{\text{mm}}{\text{s}} \times 1.5\,\text{s} = 4.5\,\text{mm}$$

Similarly, points a and c each move 3 mm, and point d moves 1.5 mm. Hence the plot at $t = 1.5$ s is



The rate of angular deformation is

$$\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = U\frac{1}{h} + 0 = \frac{U}{h} = 4\frac{\text{mm}}{\text{s}} \times \frac{1}{4\,\text{mm}} = 1\,\text{s}^{-1}$$

The rate of rotation is

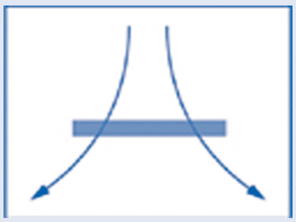
$$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) = \frac{1}{2}\left(0 - \frac{U}{h}\right) = -\frac{1}{2} \times 4\frac{\text{mm}}{\text{s}} \times \frac{1}{4\,\text{mm}} = -0.5\,\text{s}^{-1} \longleftarrow \omega_z$$

In this problem we have a viscous flow, and hence should have expected both angular deformation and particle rotation.

b. Linear Deformation

VIDEO

Linear Deformation.



During linear deformation, the shape of the fluid element, described by the angles at its vertices, remains unchanged, since all right angles continue to be right angles (see Fig. 5.5). The element will change length in the x direction only if $\partial u/\partial x$ is other than zero. Similarly, a change in the y dimension requires a nonzero value of $\partial v/\partial y$ and a change in the z dimension requires a nonzero value of $\partial w/\partial z$. These quantities represent the components of longitudinal rates of strain in the x, y, and z directions, respectively.

Changes in length of the sides may produce changes in volume of the element. The rate of local instantaneous *volume dilation* is given by

$$\text{Volume dilation rate} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = \nabla \cdot \vec{V} \tag{5.20}$$

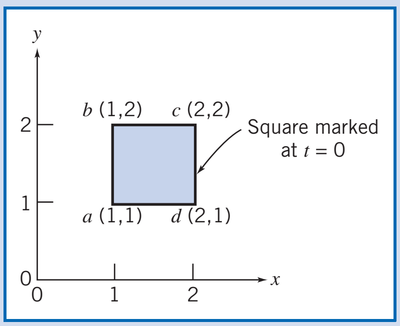
For incompressible flow, the rate of volume dilation is zero (Eq. 5.1c).

Example 5.8 DEFORMATION RATES FOR FLOW IN A CORNER

The velocity field $\vec{V} = Ax\hat{i} - Ay\hat{j}$ represents flow in a "corner," as shown in Example 5.4, where $A = 0.3\,\text{s}^{-1}$ and the coordinates are measured in meters. A square is marked in the fluid as shown at $t = 0$. Evaluate the new positions of the four corner points when point a has moved to $x = \frac{3}{2}$ m after τ seconds. Evaluate the rates of linear deformation in the x and y directions. Compare area $a'b'c'd'$ at $t = \tau$ with area $abcd$ at $t = 0$. Comment on the significance of this result.

Given: $\vec{V} = Ax\hat{i} - Ay\hat{j}$; $A = 0.3\,\text{s}^{-1}$, x and y in meters.

Find: (a) Position of square at $t = \tau$ when a is at a' at $x = \frac{3}{2}$ m.
(b) Rates of linear deformation.
(c) Area $a'b'c'd'$ compared with area $abcd$.
(d) Significance of the results.



Solution:
First we must find τ, so we must follow a fluid particle using a Lagrangian description. Thus

$$u = \frac{dx_p}{dt} = Ax_p, \quad \frac{dx}{x} = A\,dt, \text{ so } \int_{x_0}^{x}\frac{dx}{x} = \int_0^{\tau} A\,dt \quad \text{and} \quad \ln\frac{x}{x_0} = A\tau$$

$$\tau = \frac{\ln x/x_0}{A} = \frac{\ln\left(\frac{3}{2}\right)}{0.3\,\text{s}^{-1}} = 1.35\,\text{s}$$

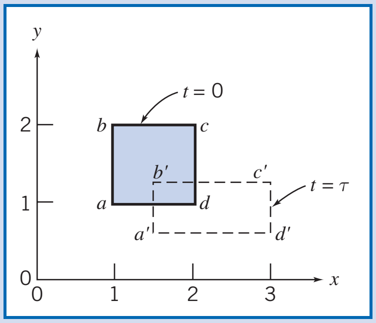
In the y direction

$$v = \frac{dy_p}{dt} = -Ay_p \qquad \frac{dy}{y} = -A\,dt \qquad \frac{y}{y_0} = e^{-A\tau}$$

The point coordinates at τ are:

Point	$t = 0$	$t = \tau$
a	(1, 1)	$\left(\frac{3}{2}, \frac{2}{3}\right)$
b	(1, 2)	$\left(\frac{3}{2}, \frac{4}{3}\right)$
c	(2, 2)	$\left(3, \frac{4}{3}\right)$
d	(2, 1)	$\left(3, \frac{2}{3}\right)$

The plot is:



Notes:

- ✓ Parallel planes remain parallel; there is linear deformation but no angular deformation.
- ✓ The flow is irrotational ($\partial v/\partial x - \partial u/\partial y = 0$).
- ✓ Volume is conserved because the two rates of linear deformation are equal and opposite.
- ✓ The NCFMF video *Flow Visualization* (see http://web.mit.edu/fluids/www/Shapiro/ncfmf.html for free online viewing of this film) uses hydrogen bubble time-streak markers to demonstrate experimentally that the area of a marked fluid square is conserved in two-dimensional incompressible flow.

The *Excel* workbook for this problem shows an animation of this motion.

The rates of linear deformation are:

$$\frac{\partial u}{\partial x} = \frac{\partial}{\partial x}Ax = A = 0.3\ \text{s}^{-1} \qquad \text{in the } x \text{ direction}$$

$$\frac{\partial v}{\partial y} = \frac{\partial}{\partial y}(-Ay) = -A = -0.3\ \text{s}^{-1} \qquad \text{in the } y \text{ direction}$$

The rate of volume dilation is

$$\nabla \cdot \vec{V} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = A - A = 0$$

Area $abcd = 1\ \text{m}^2$ and area $a'b'c'd' = \left(3 - \frac{3}{2}\right)\left(\frac{4}{3} - \frac{2}{3}\right) = 1\ \text{m}^2$.

We have shown in this section that the velocity field can be used to find the acceleration, rotation, angular deformation, and linear deformation of a fluid particle in a flow field.

Momentum Equation 5.4

A dynamic equation describing fluid motion may be obtained by applying Newton's second law to a particle. To derive the differential form of the momentum equation, we shall apply Newton's second law to an infinitesimal fluid particle of mass dm.

Recall that Newton's second law for a finite system is given by

$$\vec{F} = \left.\frac{d\vec{P}}{dt}\right)_{\text{system}} \tag{4.2a}$$

where the linear momentum, $\vec{P}$, of the system is given by

$$\vec{P}_{\text{system}} = \int_{\text{mass (system)}} \vec{V}\,dm \tag{4.2b}$$

Then, for an infinitesimal system of mass dm, Newton's second law can be written

$$d\vec{F} = dm \left.\frac{d\vec{V}}{dt}\right)_{\text{system}} \tag{5.21}$$

Having obtained an expression for the acceleration of a fluid element of mass dm, moving in a velocity field (Eq. 5.9), we can write Newton's second law as the vector equation

$$d\vec{F} = dm\frac{D\vec{V}}{Dt} = dm\left[u\frac{\partial\vec{V}}{\partial x} + v\frac{\partial\vec{V}}{\partial y} + w\frac{\partial\vec{V}}{\partial z} + \frac{\partial\vec{V}}{\partial t}\right] \tag{5.22}$$

We now need to obtain a suitable formulation for the force, $d\vec{F}$, or its components, dF_x, dF_y, and dF_z, acting on the element.

Forces Acting on a Fluid Particle

Recall that the forces acting on a fluid element may be classified as body forces and surface forces; surface forces include both normal forces and tangential (shear) forces.

We shall consider the x component of the force acting on a differential element of mass dm and volume $d\mathcal{V} = dx\,dy\,dz$. Only those stresses that act in the x direction will give rise to surface forces in the x direction. If the stresses at the center of the differential element are taken to be σ_{xx}, τ_{yx}, and τ_{zx}, then the stresses acting in the x direction on all faces of the element (obtained by a Taylor series expansion about the center of the element) are as shown in Fig. 5.9.

To obtain the net surface force in the x direction, dF_{S_x}, we must sum the forces in the x direction. Thus,

$$\begin{aligned} dF_{S_x} = &\left(\sigma_{xx} + \frac{\partial\sigma_{xx}}{\partial x}\frac{dx}{2}\right)dy\,dz - \left(\sigma_{xx} - \frac{\partial\sigma_{xx}}{\partial x}\frac{dx}{2}\right)dy\,dz \\ &+ \left(\tau_{yx} + \frac{\partial\tau_{yx}}{\partial y}\frac{dy}{2}\right)dx\,dz - \left(\tau_{yx} - \frac{\partial\tau_{yx}}{\partial y}\frac{dy}{2}\right)dx\,dz \\ &+ \left(\tau_{zx} + \frac{\partial\tau_{zx}}{\partial z}\frac{dz}{2}\right)dx\,dy - \left(\tau_{zx} - \frac{\partial\tau_{zx}}{\partial z}\frac{dz}{2}\right)dx\,dy \end{aligned}$$

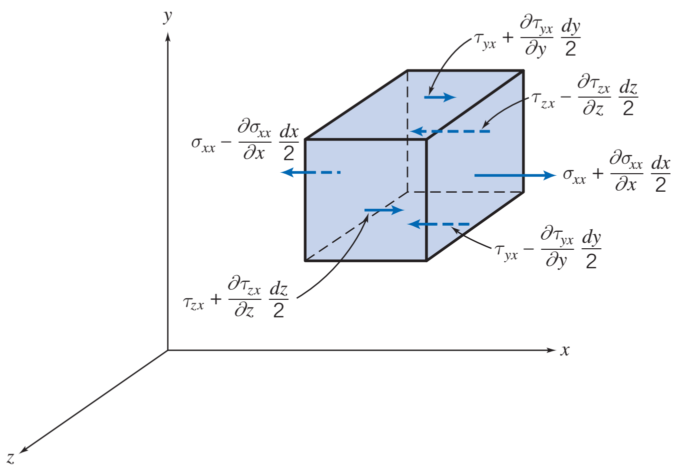


Fig. 5.9 Stresses in the x direction on an element of fluid.

On simplifying, we obtain

$$dF_{S_x} = \left(\frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \right) dx\, dy\, dz$$

When the force of gravity is the only body force acting, then the body force per unit mass is $\vec{g}$. The net force in the x direction, dF_x, is given by

$$dF_x = dF_{B_x} + dF_{S_x} = \left(\rho g_x + \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \right) dx\, dy\, dz \tag{5.23a}$$

We can derive similar expressions for the force components in the y and z directions:

$$dF_y = dF_{B_y} + dF_{S_y} = \left(\rho g_y + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} \right) dx\, dy\, dz \tag{5.23b}$$

$$dF_z = dF_{B_z} + dF_{S_z} = \left(\rho g_z + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_{zz}}{\partial z} \right) dx\, dy\, dz \tag{5.23c}$$

Differential Momentum Equation

We have now formulated expressions for the components, dF_x, dF_y, and dF_z, of the force, $d\vec{F}$, acting on the element of mass dm. If we substitute these expressions (Eqs. 5.23) for the force components into the x, y, and z components of Eq. 5.22, we obtain the differential equations of motion,

$$\rho g_x + \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} = \rho \left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} \right) \tag{5.24a}$$

$$\rho g_y + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} = \rho \left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} \right) \tag{5.24b}$$

$$\rho g_z + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_{zz}}{\partial z} = \rho \left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} \right) \tag{5.24c}$$

Equations 5.24 are the differential equations of motion for any fluid satisfying the continuum assumption. Before the equations can be used to solve for u, v, and w, suitable expressions for the stresses must be obtained in terms of the velocity and pressure fields.

Newtonian Fluid: Navier–Stokes Equations

For a Newtonian fluid the viscous stress is directly proportional to the rate of shearing strain (angular deformation rate). We saw in Chapter 2 that for one-dimensional laminar Newtonian flow the shear stress is proportional to the rate of angular deformation: $\tau_{yx} = du/dy$ (Eq. 2.15). For a three-dimensional flow the situation is a bit more complicated (among other things we need to use the more complicated expressions for rate of angular deformation, Eq. 5.19). The stresses may be expressed in terms of velocity gradients and fluid properties in rectangular coordinates as follows:[6]

[6]The derivation of these results is beyond the scope of this book. Detailed derivations may be found in Daily and Harleman [2], Schlichting [3], and White [4].

$$\tau_{xy} = \tau_{yx} = \mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right) \tag{5.25a}$$

$$\tau_{yz} = \tau_{zy} = \mu\left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right) \tag{5.25b}$$

$$\tau_{zx} = \tau_{xz} = \mu\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right) \tag{5.25c}$$

$$\sigma_{xx} = -p - \frac{2}{3}\mu\nabla\cdot\vec{V} + 2\mu\frac{\partial u}{\partial x} \tag{5.25d}$$

$$\sigma_{yy} = -p - \frac{2}{3}\mu\nabla\cdot\vec{V} + 2\mu\frac{\partial v}{\partial y} \tag{5.25e}$$

$$\sigma_{zz} = -p - \frac{2}{3}\mu\nabla\cdot\vec{V} + 2\mu\frac{\partial w}{\partial z} \tag{5.25f}$$

where p is the local thermodynamic pressure.[7] Thermodynamic pressure is related to the density and temperature by the thermodynamic relation usually called the equation of state.

If these expressions for the stresses are introduced into the differential equations of motion (Eqs. 5.24), we obtain

$$\begin{aligned}\rho\frac{Du}{Dt} = \rho g_x - \frac{\partial p}{\partial x} &+ \frac{\partial}{\partial x}\left[\mu\left(2\frac{\partial u}{\partial x} - \frac{2}{3}\nabla\cdot\vec{V}\right)\right] + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right]\\ &+ \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right]\end{aligned} \tag{5.26a}$$

$$\begin{aligned}\rho\frac{Dv}{Dt} = \rho g_y - \frac{\partial p}{\partial y} &+ \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right] + \frac{\partial}{\partial y}\left[\mu\left(2\frac{\partial v}{\partial y} - \frac{2}{3}\nabla\cdot\vec{V}\right)\right]\\ &+ \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)\right]\end{aligned} \tag{5.26b}$$

$$\begin{aligned}\rho\frac{Dw}{Dt} = \rho g_z - \frac{\partial p}{\partial z} &+ \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right] + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial w}{\partial z} + \frac{\partial w}{\partial y}\right)\right]\\ &+ \frac{\partial}{\partial z}\left[\mu\left(2\frac{\partial w}{\partial z} - \frac{2}{3}\nabla\cdot\vec{V}\right)\right]\end{aligned} \tag{5.26c}$$

These equations of motion are called the *Navier–Stokes* equations. The equations are greatly simplified when applied to *incompressible flow* with *constant viscosity*. Under these conditions the equations reduce to

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = \rho g_x - \frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \tag{5.27a}$$

[7]Sabersky et al. [5] discuss the relation between the thermodynamic pressure and the average pressure defined as $p = -(\sigma_{xx} + \sigma_{yy} + \sigma_{zz})/3$.

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=\rho g_y-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right) \tag{5.27b}$$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=\rho g_z-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \tag{5.27c}$$

This form of the Navier–Stokes equations is probably (next to the Bernoulli equation) the most famous set of equations in fluid mechanics, and has been widely studied. These equations, with the continuity equation (Eq. 5.1c), form a set of four coupled nonlinear partial differential equations for u, v, w, and p. In principle, these four equations describe many common flows; the only restrictions are that the fluid be Newtonian (with a constant viscosity) and incompressible. For example, lubrication theory (describing the behavior of machine bearings), pipe flows, and even the motion of your coffee as you stir it are explained by these equations. Unfortunately, they are impossible to solve analytically, except for the most basic cases [3], in which we have simple boundaries and initial or boundary conditions! We will solve the equations for such a simple problem in Example 5.9.

The Navier–Stokes equations for constant density and viscosity are given in cylindrical coordinates in Appendix B; they have also been derived for spherical coordinates [3]. We will apply the cylindrical coordinate form in solving Example 5.10.

In recent years computational fluid dynamics (CFD) computer applications (such as *Fluent* [6] and *STAR-CD* [7]) have been developed for analyzing the Navier–Stokes equations for more complicated, real-world problems. Although a detailed treatment of the topic is beyond the scope of this text, we shall have a brief introduction to CFD in the next section.

For the case of frictionless flow ($\mu = 0$) the equations of motion (Eqs. 5.26 or Eqs. 5.27) reduce to *Euler's equation*,

$$\rho\frac{D\vec{V}}{Dt}=\rho\vec{g}-\nabla p$$

We shall consider the case of frictionless flow in Chapter 6.

Example 5.9 ANALYSIS OF FULLY DEVELOPED LAMINAR FLOW DOWN AN INCLINED PLANE SURFACE

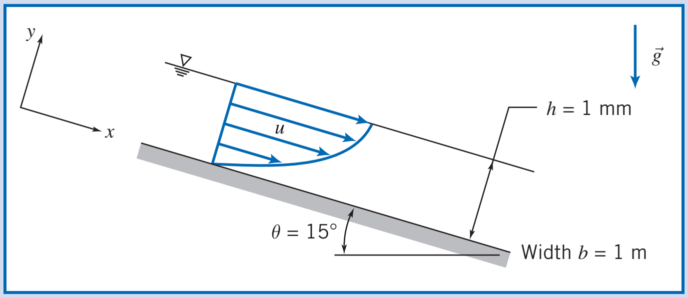
A liquid flows down an inclined plane surface in a steady, fully developed laminar film of thickness h. Simplify the continuity and Navier–Stokes equations to model this flow field. Obtain expressions for the liquid velocity profile, the shear stress distribution, the volume flow rate, and the average velocity. Relate the liquid film thickness to the volume flow rate per unit depth of surface normal to the flow. Calculate the volume flow rate in a film of water $h = 1$ mm thick, flowing on a surface $b = 1$ m wide, inclined at $\theta = 15°$ to the horizontal.

Given: Liquid flow down an inclined plane surface in a steady, fully developed laminar film of thickness h.

Find: (a) Continuity and Navier–Stokes equations simplified to model this flow field.
(b) Velocity profile.
(c) Shear stress distribution.
(d) Volume flow rate per unit depth of surface normal to diagram.
(e) Average flow velocity.
(f) Film thickness in terms of volume flow rate per unit depth of surface normal to diagram.
(g) Volume flow rate in a film of water 1 mm thick on a surface 1 m wide, inclined at 15° to the horizontal.

Solution:

The geometry and coordinate system used to model the flow field are shown. (It is convenient to align one coordinate with the flow down the plane surface.)



The governing equations written for incompressible flow with constant viscosity are

$$\cancelto{4}{\frac{\partial u}{\partial x}} + \frac{\partial v}{\partial y} + \cancelto{3}{\frac{\partial w}{\partial z}} = 0 \qquad (5.1c)$$

$$\rho\left(\cancelto{1}{\frac{\partial u}{\partial t}} + u\cancelto{4}{\frac{\partial u}{\partial x}} + \cancelto{5}{v\frac{\partial u}{\partial y}} + \cancelto{3}{w\frac{\partial u}{\partial z}}\right) = \rho g_x - \cancelto{4}{\frac{\partial p}{\partial x}} + \mu\left(\cancelto{4}{\frac{\partial^2 u}{\partial x^2}} + \frac{\partial^2 u}{\partial y^2} + \cancelto{3}{\frac{\partial^2 u}{\partial z^2}}\right) \qquad (5.27a)$$

$$\rho\left(\cancelto{1}{\frac{\partial v}{\partial t}} + u\cancelto{4}{\frac{\partial v}{\partial x}} + v\cancelto{5}{\frac{\partial v}{\partial y}} + w\cancelto{3}{\frac{\partial v}{\partial z}}\right) = \rho g_y - \frac{\partial p}{\partial y} + \mu\left(\cancelto{4}{\frac{\partial^2 v}{\partial x^2}} + \cancelto{5}{\frac{\partial^2 v}{\partial y^2}} + \cancelto{3}{\frac{\partial^2 v}{\partial z^2}}\right) \qquad (5.27b)$$

$$\rho\left(\cancelto{1}{\frac{\partial w}{\partial t}} + u\cancelto{3}{\frac{\partial w}{\partial x}} + v\cancelto{3}{\frac{\partial w}{\partial y}} + w\cancelto{3}{\frac{\partial w}{\partial z}}\right) = \cancelto{3}{\rho g_z} - \frac{\partial p}{\partial z} + \mu\left(\cancelto{3}{\frac{\partial^2 w}{\partial x^2}} + \cancelto{3}{\frac{\partial^2 w}{\partial y^2}} + \cancelto{3}{\frac{\partial^2 w}{\partial z^2}}\right) \qquad (5.27c)$$

The terms canceled to simplify the basic equations are keyed by number to the assumptions listed below. The assumptions are discussed in the order in which they are applied to simplify the equations.

Assumptions: (1) Steady flow (given).
(2) Incompressible flow; $\rho = \text{constant}$.
(3) No flow or variation of properties in the z direction; $w = 0$ and $\partial/\partial z = 0$.
(4) Fully developed flow, so no properties vary in the x direction; $\partial/\partial x = 0$.

Assumption (1) eliminates time variations in any fluid property.
Assumption (2) eliminates space variations in density.
Assumption (3) states that there is no z component of velocity and no property variations in the z direction. All terms in the z component of the Navier–Stokes equation cancel.
After assumption (4) is applied, the continuity equation reduces to $\partial v/\partial y = 0$. Assumptions (3) and (4) also indicate that $\partial v/\partial z = 0$ and $\partial v/\partial x = 0$. Therefore v must be constant. Since v is zero at the solid surface, then v must be zero everywhere.
The fact that $v = 0$ reduces the Navier–Stokes equations further, as indicated by (5) in Eqs 5.27a and 5.27b. The final simplified equations are

$$0 = \rho g_x + \mu\frac{\partial^2 u}{\partial y^2} \qquad (1)$$

$$0 = \rho g_y - \frac{\partial p}{\partial y} \qquad (2)$$

Since $\partial u/\partial z = 0$ (assumption 3) and $\partial u/\partial x = 0$ (assumption 4), then u is at most a function of y, and $\partial^2 u/\partial y^2 = d^2u/dy^2$, and from Eq. 1, then

$$\frac{d^2u}{dy^2} = -\frac{\rho g_x}{\mu} = -\rho g\frac{\sin\theta}{\mu}$$

Integrating,

$$\frac{du}{dy} = -\rho g \frac{\sin\theta}{\mu} y + c_1 \tag{3}$$

and integrating again,

$$u = -\rho g \frac{\sin\theta}{\mu}\frac{y^2}{2} + c_1 y + c_2 \tag{4}$$

The boundary conditions needed to evaluate the constants are the no-slip condition at the solid surface ($u = 0$ at $y = 0$) and the zero-shear-stress condition at the liquid free surface ($du/dy = 0$ at $y = h$).

Evaluating Eq. 4 at $y = 0$ gives $c_2 = 0$. From Eq. 3 at $y = h$,

$$0 = -\rho g \frac{\sin\theta}{\mu} h + c_1$$

or

$$c_1 = \rho g \frac{\sin\theta}{\mu} h$$

Substituting into Eq. 4 we obtain the velocity profile

$$u = -\rho g \frac{\sin\theta}{\mu}\frac{y^2}{2} + \rho g \frac{\sin\theta}{\mu} hy$$

or

$$u = \rho g \frac{\sin\theta}{\mu}\left(hy - \frac{y^2}{2}\right) \longleftarrow u(y)$$

The shear stress distribution is (from Eq. 5.25a after setting $\partial v/\partial x$ to zero, or alternatively, for one-dimensional flow, from Eq. 2.15)

$$\tau_{yx} = \mu \frac{du}{dy} = \rho g \sin\theta\,(h - y) \longleftarrow \tau_{yx}(y)$$

The shear stress in the fluid reaches its maximum value at the wall ($y = 0$); as we expect, it is zero at the free surface ($y = h$). At the wall the shear stress τ_{yx} is positive but the surface normal *for the fluid* is in the negative y direction; hence the shear force acts in the negative x direction, and just balances the x component of the body force acting on the fluid. The volume flow rate is

$$Q = \int_A u\,dA = \int_0^h u\,b\,dy$$

where b is the surface width in the z direction. Substituting,

$$Q = \int_0^h \frac{\rho g \sin\theta}{\mu}\left(hy - \frac{y^2}{2}\right) b\,dy = \rho g \frac{\sin\theta\,b}{\mu}\left[\frac{hy^2}{2} - \frac{y^3}{6}\right]_0^h$$

$$Q = \frac{\rho g \sin\theta\,b}{\mu}\frac{h^3}{3} \longleftarrow (5)\,Q$$

The average flow velocity is $\overline{V} = Q/A = Q/bh$. Thus

$$\overline{V} = \frac{Q}{bh} = \frac{\rho g \sin\theta}{\mu}\frac{h^2}{3} \longleftarrow \overline{V}$$

Solving for film thickness gives

$$h = \left[\frac{3\mu Q}{\rho g \sin\theta\, b}\right]^{1/3} \qquad (6)\; h$$

A film of water $h = 1$ mm thick on a plane $b = 1$ m wide, inclined at $\theta = 15°$, would carry

$$Q = 999\,\frac{\text{kg}}{\text{m}^3} \times 9.81\,\frac{\text{m}}{\text{s}^2} \times \sin(15°) \times 1\,\text{m} \times \frac{\text{m}\cdot\text{s}}{1.00\times 10^{-3}\,\text{kg}}$$

$$\times\ \frac{(0.001)^3\,\text{m}^3}{3} \times 1000\,\frac{\text{L}}{\text{m}^3}$$

$$Q = 0.846\ \text{L/s} \qquad Q$$

Notes:

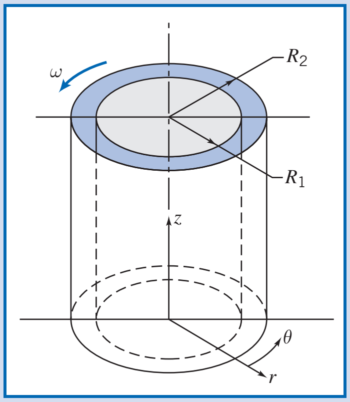
- ✓ This problem illustrates how the full Navier–Stokes equations (Eqs. 5.27) can sometimes be reduced to a set of solvable equations (Eqs. 1 and 2 in this problem).
- ✓ After integration of the simplified equations, boundary (or initial) conditions are used to complete the solution.
- ✓ Once the velocity field is obtained, other useful quantities (e.g., shear stress, volume flow rate) can be found.
- ✓ Equations (5) and (6) show that even for fairly simple problems the results can be quite complicated: The depth of the flow depends in a nonlinear way on flow rate ($h \propto Q^{1/3}$).

Example 5.10 ANALYSIS OF LAMINAR VISCOMETRIC FLOW BETWEEN COAXIAL CYLINDERS

A viscous liquid fills the annular gap between vertical concentric cylinders. The inner cylinder is stationary, and the outer cylinder rotates at constant speed. The flow is laminar. Simplify the continuity, Navier–Stokes, and tangential shear stress equations to model this flow field. Obtain expressions for the liquid velocity profile and the shear stress distribution. Compare the shear stress at the surface of the inner cylinder with that computed from a planar approximation obtained by "unwrapping" the annulus into a plane and assuming a linear velocity profile across the gap. Determine the ratio of cylinder radii for which the planar approximation predicts the correct shear stress at the surface of the inner cylinder within 1 percent.

Given: Laminar viscometric flow of liquid in annular gap between vertical concentric cylinders. The inner cylinder is stationary, and the outer cylinder rotates at constant speed.

Find:
(a) Continuity and Navier–Stokes equations simplified to model this flow field.
(b) Velocity profile in the annular gap.
(c) Shear stress distribution in the annular gap.
(d) Shear stress at the surface of the inner cylinder.
(e) Comparison with "planar" approximation for constant shear stress in the narrow gap between cylinders.
(f) Ratio of cylinder radii for which the planar approximation predicts shear stress within 1 percent of the correct value.



Solution:
The geometry and coordinate system used to model the flow field are shown. (The z coordinate is directed vertically upward; as a consequence, $g_r = g_\theta = 0$ and $g_z = -g$.)

The continuity, Navier–Stokes, and tangential shear stress equations (from Appendix B) written for incompressible flow with constant viscosity are

$$\frac{1}{r}\frac{\partial}{\partial r}(rv_r) + \frac{1}{r}\cancelto{4}{\frac{\partial}{\partial\theta}}(v_\theta) + \frac{\partial}{\partial z}\cancelto{3}{(v_z)} = 0 \tag{B.1}$$

r component:

$$\rho\left(\cancelto{1}{\frac{\partial v_r}{\partial t}} + v_r\cancelto{5}{\frac{\partial v_r}{\partial r}} + \frac{v_\theta}{r}\cancelto{4}{\frac{\partial v_r}{\partial\theta}} - \frac{v_\theta^2}{r} + v_z\cancelto{3}{\frac{\partial v_r}{\partial z}}\right)$$
$$= \rho\cancelto{0}{g_r} - \frac{\partial p}{\partial r} + \mu\left\{\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}\cancelto{5}{[rv_r]}\right) + \frac{1}{r^2}\cancelto{4}{\frac{\partial^2 v_r}{\partial\theta^2}} - \frac{2}{r^2}\cancelto{4}{\frac{\partial v_\theta}{\partial\theta}} + \cancelto{3}{\frac{\partial^2 v_r}{\partial z^2}}\right\} \tag{B.3a}$$

θ component:

$$\rho\left(\cancelto{1}{\frac{\partial v_\theta}{\partial t}} + \cancelto{5}{v_r}\frac{\partial v_\theta}{\partial r} + \frac{v_\theta}{r}\cancelto{4}{\frac{\partial v_\theta}{\partial\theta}} + \cancelto{5}{\frac{v_r v_\theta}{r}} + v_z\cancelto{3}{\frac{\partial v_\theta}{\partial z}}\right)$$
$$= \rho\cancelto{0}{g_\theta} - \frac{1}{r}\cancelto{4}{\frac{\partial p}{\partial\theta}} + \mu\left\{\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}[rv_\theta]\right) + \frac{1}{r^2}\cancelto{4}{\frac{\partial^2 v_\theta}{\partial\theta^2}} + \frac{2}{r^2}\cancelto{4}{\frac{\partial v_r}{\partial\theta}} + \cancelto{3}{\frac{\partial^2 v_\theta}{\partial z^2}}\right\} \tag{B.3b}$$

z component:

$$\rho\left(\cancelto{1}{\frac{\partial v_z}{\partial t}} + \cancelto{5}{v_r}\frac{\partial v_z}{\partial r} + \frac{v_\theta}{r}\cancelto{4}{\frac{\partial v_z}{\partial\theta}} + v_z\cancelto{3}{\frac{\partial v_z}{\partial z}}\right) = \rho g_z - \frac{\partial p}{\partial z} + \mu\left\{\frac{1}{r}\frac{\partial}{\partial r}\cancelto{3}{\left(r\frac{\partial v_z}{\partial r}\right)} + \frac{1}{r^2}\cancelto{3}{\frac{\partial^2 v_z}{\partial\theta^2}} + \cancelto{3}{\frac{\partial^2 v_z}{\partial z^2}}\right\} \tag{B.3c}$$

$$\tau_{r\theta} = \mu\left[r\frac{\partial}{\partial r}\left(\frac{v_\theta}{r}\right) + \frac{1}{r}\cancelto{4}{\frac{\partial v_r}{\partial\theta}}\right] \tag{B.2}$$

The terms canceled to simplify the basic equations are keyed by number to the assumptions listed below. The assumptions are discussed in the order in which they are applied to simplify the equations.

Assumptions: (1) Steady flow; angular speed of outer cylinder is constant.
(2) Incompressible flow; $\rho = \text{constant}$.
(3) No flow or variation of properties in the z direction; $v_z = 0$ and $\partial/\partial z = 0$.
(4) Circumferentially symmetric flow, so properties do not vary with θ, so $\partial/\partial\theta = 0$.

Assumption (1) eliminates time variations in fluid properties.

Assumption (2) eliminates space variations in density.

Assumption (3) causes all terms in the z component of the Navier–Stokes equation (Eq. B.3c) to cancel, except for the hydrostatic pressure distribution.

After assumptions (3) and (4) are applied, the continuity equation (Eq. B.1) reduces to

$$\frac{1}{r}\frac{\partial}{\partial r}(rv_r) = 0$$

Because $\partial/\partial\theta = 0$ and $\partial/\partial z = 0$ by assumptions (3) and (4), then $\dfrac{\partial}{\partial r} \to \dfrac{d}{dr}$, so integrating gives

$$rv_r = \text{constant}$$

Since v_r is zero at the solid surface of each cylinder, then v_r must be zero everywhere.

The fact that $v_r = 0$ reduces the Navier–Stokes equations further, as indicated by cancellations (5). The final equations (Eqs. B.3a and B.3b) reduce to

$$-\rho\frac{v_\theta^2}{r} = -\frac{\partial p}{\partial r}$$

$$0 = \mu\left\{\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}[rv_\theta]\right)\right\}$$

But since $\partial/\partial\theta = 0$ and $\partial/\partial z = 0$ by assumptions (3) and (4), then v_θ is a function of radius only, and

$$\frac{d}{dr}\left(\frac{1}{r}\frac{d}{dr}[rv_\theta]\right) = 0$$

Integrating once,

$$\frac{1}{r}\frac{d}{dr}[rv_\theta] = c_1$$

or

$$\frac{d}{dr}[rv_\theta] = c_1 r$$

Integrating again,

$$rv_\theta = c_1\frac{r^2}{2} + c_2 \qquad \text{or} \qquad v_\theta = c_1\frac{r}{2} + c_2\frac{1}{r}$$

Two boundary conditions are needed to evaluate constants c_1 and c_2. The boundary conditions are

$$\begin{aligned} v_\theta &= \omega R_2 \quad &\text{at} \quad r &= R_2 \quad \text{and} \\ v_\theta &= 0 \quad &\text{at} \quad r &= R_1 \end{aligned}$$

Substituting

$$\omega R_2 = c_1\frac{R_2}{2} + c_2\frac{1}{R_2}$$

$$0 = c_1\frac{R_1}{2} + c_2\frac{1}{R_1}$$

After considerable algebra

$$c_1 = \frac{2\omega}{1-\left(\frac{R_1}{R_2}\right)^2} \quad \text{and} \quad c_2 = \frac{-\omega R_1^2}{1-\left(\frac{R_1}{R_2}\right)^2}$$

Substituting into the expression for v_θ,

$$v_\theta = \frac{\omega r}{1-\left(\frac{R_1}{R_2}\right)^2} - \frac{\omega R_1^2/r}{1-\left(\frac{R_1}{R_2}\right)^2} = \frac{\omega R_1}{1-\left(\frac{R_1}{R_2}\right)^2}\left[\frac{r}{R_1} - \frac{R_1}{r}\right] \longleftarrow v_\theta(r)$$

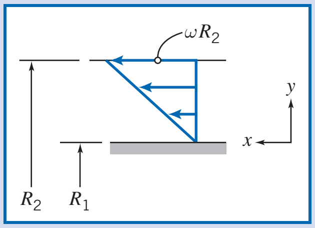
The shear stress distribution is obtained from Eq. B.2 after using assumption (4):

$$\tau_{r\theta} = \mu r \frac{d}{dr}\left(\frac{v_\theta}{r}\right) = \mu r \frac{d}{dr}\left\{\frac{\omega R_1}{1-\left(\frac{R_1}{R_2}\right)^2}\left[\frac{1}{R_1} - \frac{R_1}{r^2}\right]\right\} = \mu r \frac{\omega R_1}{1-\left(\frac{R_1}{R_2}\right)^2}(-2)\left(-\frac{R_1}{r^3}\right)$$

$$\tau_{r\theta} = \mu \frac{2\omega R_1^2}{1-\left(\frac{R_1}{R_2}\right)^2}\frac{1}{r^2} \longleftarrow \underline{\tau_{r\theta}}$$

At the surface of the inner cylinder, $r = R_1$, so

$$\tau_{\text{surface}} = \mu \frac{2\omega}{1-\left(\frac{R_1}{R_2}\right)^2} \longleftarrow \underline{\tau_{\text{surface}}}$$



For a "planar" gap

$$\tau_{\text{planar}} = \mu \frac{\Delta v}{\Delta y} = \mu \frac{\omega R_2}{R_2 - R_1}$$

or

$$\tau_{\text{planar}} = \mu \frac{\omega}{1 - \frac{R_1}{R_2}} \longleftarrow \underline{\tau_{\text{planar}}}$$

Factoring the denominator of the exact expression for shear stress at the surface gives

$$\tau_{\text{surface}} = \mu \frac{2\omega}{\left(1-\frac{R_1}{R_2}\right)\left(1+\frac{R_1}{R_2}\right)} = \mu \frac{\omega}{1-\frac{R_1}{R_2}} \cdot \frac{2}{1+\frac{R_1}{R_2}}$$

Thus

$$\frac{\tau_{\text{surface}}}{\tau_{\text{planar}}} = \frac{2}{1+\frac{R_1}{R_2}}$$

For 1 percent accuracy,

$$1.01 = \frac{2}{1+\frac{R_1}{R_2}}$$

or

$$\frac{R_1}{R_2} = \frac{1}{1.01}(2-1.01) = 0.980 \longleftarrow \underline{\frac{R_1}{R_2}}$$

The accuracy criterion is met when the gap width is less than 2 percent of the cylinder radius.

Notes:

- ✓ This problem illustrates how the full Navier–Stokes equations in cylindrical coordinates (Eqs. B.1 to B.3) can sometimes be reduced to a set of solvable equations.
- ✓ As in Example 5.9, after integration of the simplified equations, boundary (or initial) conditions are used to complete the solution.
- ✓ Once the velocity field is obtained, other useful quantities (in this problem, shear stress) can be found.

The *Excel* workbook for this problem compares the viscometer and linear velocity profiles. It also allows one to derive the appropriate value of the viscometer outer radius to meet a prescribed accuracy of the planar approximation. We will discuss the concentric cylinder–infinite parallel plates approximation again in Chapter 8.

*5.5 Introduction to Computational Fluid Dynamics

In this section we will discuss in a very basic manner the ideas behind *computational fluid dynamics* (CFD). We will first review some very basic ideas in numerically solving an ordinary and a partial differential equation using a spreadsheet such as *Excel*, with a couple of Examples. After studying these, the reader will be able to use the PC to numerically solve a range of simple CFD problems. Then, for those with further interest in CFD, we will review in more detail some concepts behind numerical methods, particularly CFD; this review will highlight some of the advantages and pitfalls of CFD. We will apply some of these concepts to a simple 1D model, but these concepts are so fundamental that they are applicable to almost any CFD calculation. As we apply the CFD solution procedure to the model, we'll comment on the extension to the general case. The goal is to enable the reader to apply the CFD solution procedure to simple nonlinear equations.

The Need for CFD

As discussed in Section 5.4, the equations describing fluid flow can be a bit intimidating. For example, even though we may limit ourselves to incompressible flows for which the viscosity is constant, we still end up with the following equations:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{5.1c}$$

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = \rho g_x - \frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \tag{5.27a}$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = \rho g_y - \frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) \tag{5.27b}$$

$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = \rho g_z - \frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) \tag{5.27c}$$

Equation 5.1c is the continuity equation (mass conservation) and Eqs. 5.27 are the Navier–Stokes equations (momentum), expressed in Cartesian coordinates. In principle, we can solve these equations for the velocity field $\vec{V} = \hat{i}u + \hat{j}v + \hat{k}w$ and pressure field p, given sufficient initial and boundary conditions. Note that in general, u, v, w, and p all depend on x, y, z, and t. In practice, there is no general analytic solution to these equations, for the combined effect of a number of reasons (none of which is insurmountable by itself):

1. They are coupled. The unknowns, u, v, w, and p, appear in all the equations (p is not in Eq. 5.1c) and we cannot manipulate the equations to end up with a single equation for any one of the unknowns. Hence we must solve for all unknowns simultaneously.
2. They are nonlinear. For example, in Eq. 5.27a, the convective acceleration term, $u\,\partial u/\partial x + v\,\partial u/\partial y + w\,\partial u/\partial z$, has products of u with itself as well as with v and w. The consequence of this is that we cannot take one solution to the equations and combine it with a second solution to obtain a third solution. We will see in Chapter 6 that if we can limit ourselves to frictionless flow, we *can* derive linear equations, which will allow us to do this combining procedure (you may wish to look at Table 6.3 for some beautiful examples of this).

*This section may be omitted without loss of continuity in the text material.

3. They are second-order partial differential equations. For example, in Eq. 5.27a, the viscous term, $\mu(\partial^2 u/\partial x^2 + \partial^2 u/\partial y^2 + \partial^2 u/\partial z^2)$, is second-order in u. These are obviously of a different order of complexity (no pun intended) than, say, a first-order ordinary differential equation.

These difficulties have led engineers, scientists, and mathematicians to adopt several approaches to the solution of fluid mechanics problems.

For relatively simple physical geometries and boundary or initial conditions, the equations can often be reduced to a solvable form. We saw two examples of this in Examples 5.9 and 5.10 (for cylindrical forms of the equations).

If we can neglect the viscous terms, the resulting incompressible, inviscid flow can often be successfully analyzed. This is the entire topic of Chapter 6.

Of course, most incompressible flows of interest do not have simple geometries and are not inviscid; for these, we are stuck with Eqs. 5.1c and 5.27. The only option remaining is to use numerical methods to analyze problems. It is possible to obtain approximate computer-based solutions to the equations for a variety of engineering problems. This is the main subject matter of CFD.

Applications of CFD

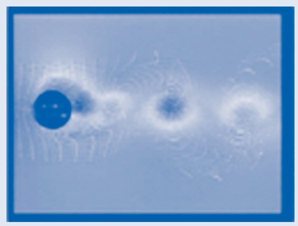
CFD is employed in a variety of applications and is now widely used in various industries. To illustrate the industrial applications of CFD, we present below some examples developed using FLUENT, a CFD software package from ANSYS, Inc. CFD is used to study the flow field around vehicles including cars, trucks, airplanes, helicopters, and ships. Figure 5.10 shows the paths taken by selected fluid particles around a Formula 1 car. By studying such pathlines and other flow attributes, engineers gain insights into how to design the car so as to reduce drag and enhance performance. The flow through a catalytic converter, a device used to clean automotive exhaust gases so that we can all breathe easier, is shown in Figure 5.11. This image shows path lines colored by velocity magnitude. CFD helps engineers develop more effective catalytic converters by allowing them to study how different chemical species mix and react in the device. Figure 5.12 presents contours of static pressure in a backward-inclined centrifugal fan used in ventilation applications.

VIDEO

Turbulent Flow in a Channel.

VIDEO

Flow Past a Cylinder.



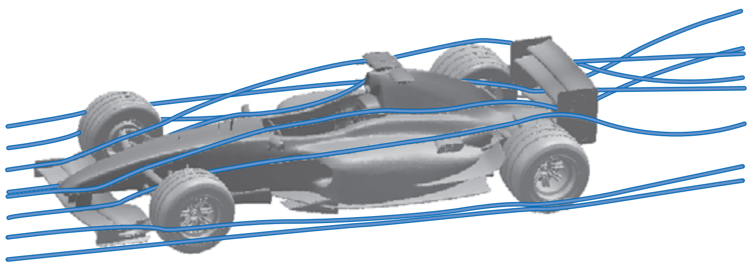


Fig. 5.10 Pathlines around a Formula 1 car. (image courtesy of ANSYS, Inc. © 2008.)

Fig. 5.11 Flow through a catalytic converter. (image courtesy of ANSYS, Inc. ©2008.)

Fig. 5.12 Static pressure contours for flow through a centrifugal fan. (image courtesy of ANSYS, Inc. © 2008.)

Fan performance characteristics obtained from the CFD simulations compared well with results from physical tests.

CFD is attractive to industry since it is more cost-effective than physical testing. However, we must note that complex flow simulations are challenging and error-prone, and it takes a lot of engineering expertise to obtain realistic solutions.

Some Basic CFD/Numerical Methods Using a Spreadsheet

Before discussing CFD in a little more detail, we can gain insight into numerical methods to solve some simple problems in fluid mechanics by using the spreadsheet. These methods will show how the student may perform elementary CFD using the PC. First, we consider solving the simplest form of a differential equation: a first-order ordinary differential equation:

$$\frac{dy}{dx} = f(x, y) \qquad y(x_0) = y_0 \tag{5.28}$$

where $f(x,y)$ is a given function. We realize that graphically the derivative dy/dx is the slope of the (as yet unknown) solution curve $y(x)$. If we are at some point (x_n, y_n) on the curve, we can follow the tangent at that point, as an approximation to actually moving along the curve itself, to find a new value for y, y_{n+1}, corresponding to a new x, x_{n+1}, as shown in Fig. 5.13. We have

$$\frac{dy}{dx} = \frac{y_{n+1} - y_n}{x_{n+1} - x_n}$$

If we choose a *step size* $h = x_{n+1} - x_n$, then the above equation can be combined with the differential equation, Eq. 5.28, to give

$$\frac{dy}{dx} = \frac{y_{n+1} - y_n}{h} = f(x_n, y_n)$$

or

$$y_{n+1} = y_n + hf(x_n, y_n) \tag{5.29a}$$

with

$$x_{n+1} = x_n + h \tag{5.29b}$$

Equations 5.29 are the basic concept behind the famous Euler method for solving a first-order ODE: A differential is replaced with a finite difference. (As we'll see in the next subsection, equations similar to Eqs. 5.29 could also have been derived more formally as the result of a truncated Taylor series.) In these equations, y_{n+1} now represents our best effort to find the next point on the solution curve. From Fig. 5.13, we see that y_{n+1} is *not* on the solution curve but close to it; if we make the triangle

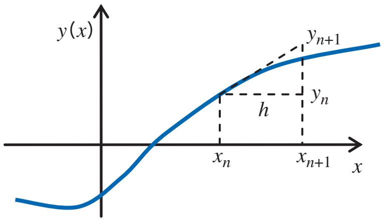


Fig. 5.13 The Euler method.

much smaller, by making the step size h smaller, then y_{n+1} will be even closer to the desired solution. We can repeatedly use the two Euler iteration equations to start at (x_0, y_0) and obtain (x_1, y_1), then (x_2, y_2), (x_3, y_3), and so on. We don't end up with an equation for the solution, but with a set of numbers; hence it is a numerical rather than an analytic method. This is the Euler method approach.

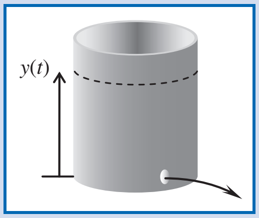
This method is very easy to set up, making it an attractive approach, but it is not very accurate: Following the tangent to a curve at each point, in an attempt to follow the curve, is pretty crude! If we make the step size h smaller, the accuracy of the method will generally increase, but obviously we then need more steps to achieve the solution. It turns out that, if we use too many steps (if h is extremely small), the accuracy of the results can actually *decrease* because, although each small step is very accurate, we will now need so many of them that round-off errors can build up. As with any numerical method, we are not guaranteed to get a solution or one that is very accurate! The Euler method is the simplest but least accurate numerical method for solving a first-order ODE; there are a number of more sophisticated ones available, as discussed in any good numerical methods text [8, 9].

Let's illustrate the method with an Example.

Example 5.11 THE EULER METHOD SOLUTION FOR DRAINING A TANK

A tank contains water at an initial depth $y_0 = 1$ m. The tank diameter is $D = 250$ mm. A hole of diameter $d = 2$ mm appears at the bottom of the tank. A reasonable model for the water level over time is

$$\frac{dy}{dt} = -\left(\frac{d}{D}\right)^2 \sqrt{2gy} \qquad y(0) = y_0$$



Using 11-point and 21-point Euler methods, estimate the water depth after $t = 100$ min, and compute the errors compared to the exact solution

$$y_{\text{exact}}(t) = \left[\sqrt{y_0} - \left(\frac{d}{D}\right)^2 \sqrt{\frac{g}{2}}\, t\right]^2$$

Plot the Euler and exact results.

Given: Water draining from a tank.

Find: Water depth after 100 min; plot of depth versus time; accuracy of results.

Solution: Use the Euler equations, Eq. (5.29).

Governing equations: $y_{n+1} = y_n + hf(t_n, y_n) \quad t_{n+1} = t_n + h$

with

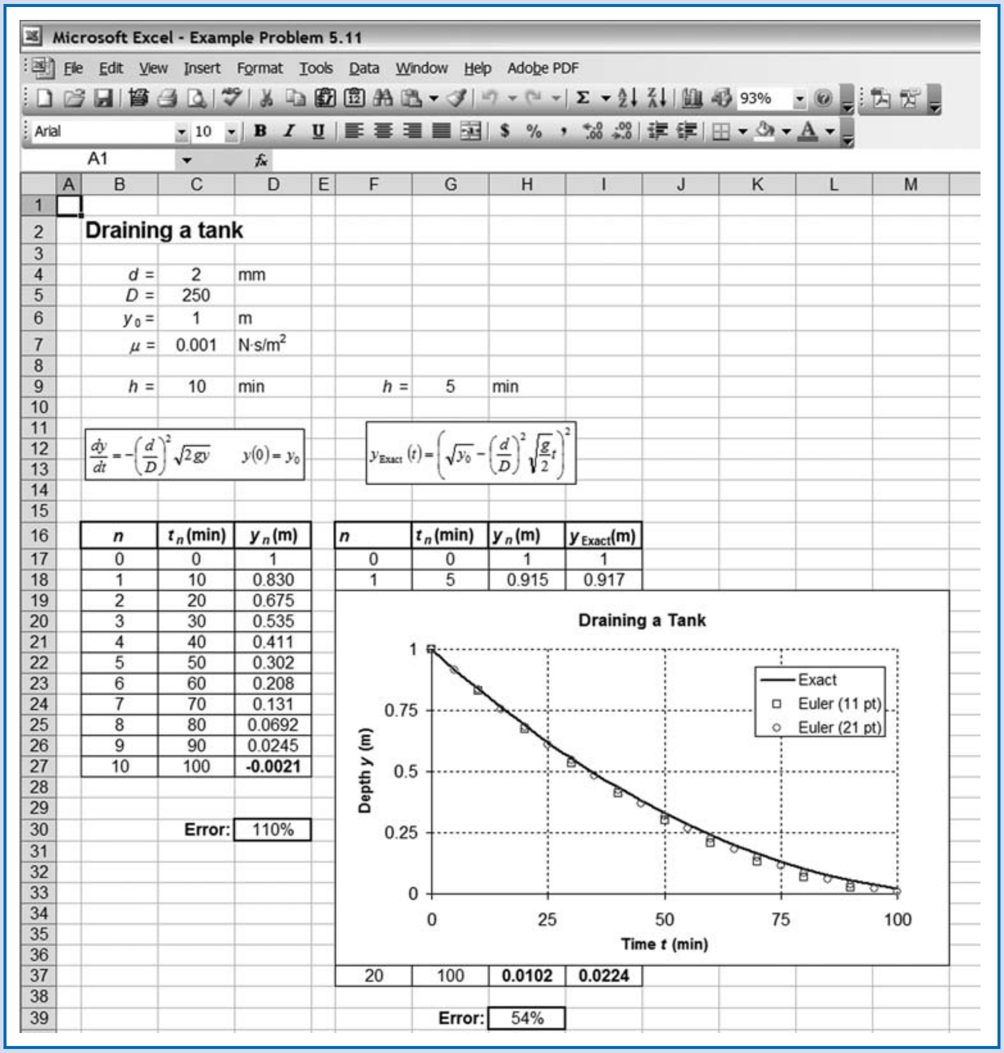
$$f(t_n, y_n) = -\left(\frac{d}{D}\right)^2 \sqrt{2gy_n} \qquad y_0 = 1$$

(Note that in using Eqs. 5.29 we use t instead of x.)

This is convenient for solving using a spreadsheet such as *Excel*, as shown below. We obtain the following results:

$$\text{Depth after 100 min} = -0.0021\,\text{m (Euler 11 point)}$$
$$= 0.0102\,\text{m (Euler 21 point)}$$
$$= 0.0224\,\text{m (Exact)} \longleftarrow y(100\,\text{min})$$

$$\text{Error after 100 min} = 110\%\,\text{(Euler 11 point)}$$
$$= 54\%\,\text{(Euler 21 point)} \longleftarrow \text{Error}$$



n	t_n (min)	y_n (m)
0	0	1
1	10	0.830
2	20	0.675
3	30	0.535
4	40	0.411
5	50	0.302
6	60	0.208
7	70	0.131
8	80	0.0692
9	90	0.0245
10	100	**-0.0021**

n	t_n (min)	y_n (m)	y_{Exact} (m)
0	0	1	1
1	5	0.915	0.917
20	100	**0.0102**	**0.0224**

This Example shows a simple application of the Euler method. Note that although the errors after 100 min are large for both Euler solutions, their plots are reasonably close to the exact solution.

The *Excel* workbook for this problem can be modified for solving a variety of fluids problems that involve first order ODEs.

Another basic application of a numerical method to a fluid mechanics problem is when we have two-dimensional, steady, incompressible, inviscid flow. These seem like a severe set of restrictions on the flow, but analysis of flows with these assumptions leads to very good predictions for real flows, for example, for the lift on a wing section. This is the topic of Chapter 6, but for now we simply state that under many circumstances such flows can be modeled with the Laplace equation,

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = 0$$

where ψ is the stream function. We leave out the steps here (they consist of approximating each differential with a Taylor series), but a numerical approximation of this equation is

$$\frac{\psi_{i+1,j} + \psi_{i-1,j}}{h^2} + \frac{\psi_{i,j+1} + \psi_{i,j-1}}{h^2} - 4\frac{\psi_{i,j}}{h^2} = 0$$

Here h is the step size in the x or y direction, and $\psi_{i,j}$ is the value of ψ at the ith value of x and jth value of y (see Fig. 5.14). Rearranging and simplifying,

$$\psi_{i,j} = \frac{1}{4}\left(\psi_{i+1,j} + \psi_{i-1,j} + \psi_{i,j+1} + \psi_{i,j-1}\right) \tag{5.30}$$

This equation indicates that the value of the stream function ψ is simply the average of its four neighbors! To use this equation, we need to specify the values of the stream function at all boundaries; Eq. 5.30 then allows computation of interior values.

Equation 5.30 is ideal for solving using a spreadsheet such as *Excel*. We again consider an Example.

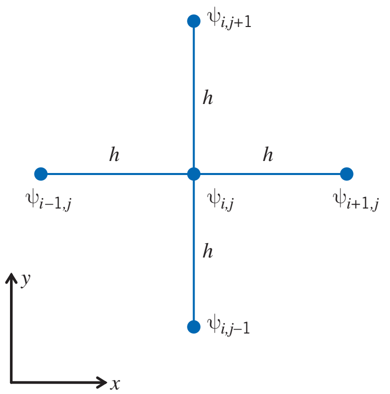
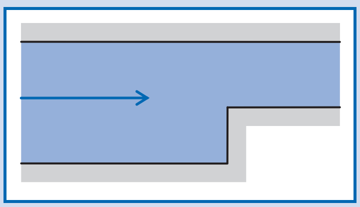


Fig. 5.14 Scheme for discretizing the Laplace equation.

Example 5.12 NUMERICAL MODELING OF FLOW OVER A CORNER

Consider a two-dimensional steady, incompressible, inviscid flow in a channel in which the area is reduced by half. Plot the streamlines.



Given: Flow in a channel in which the area is reduced by half.

Find: Streamline plot.

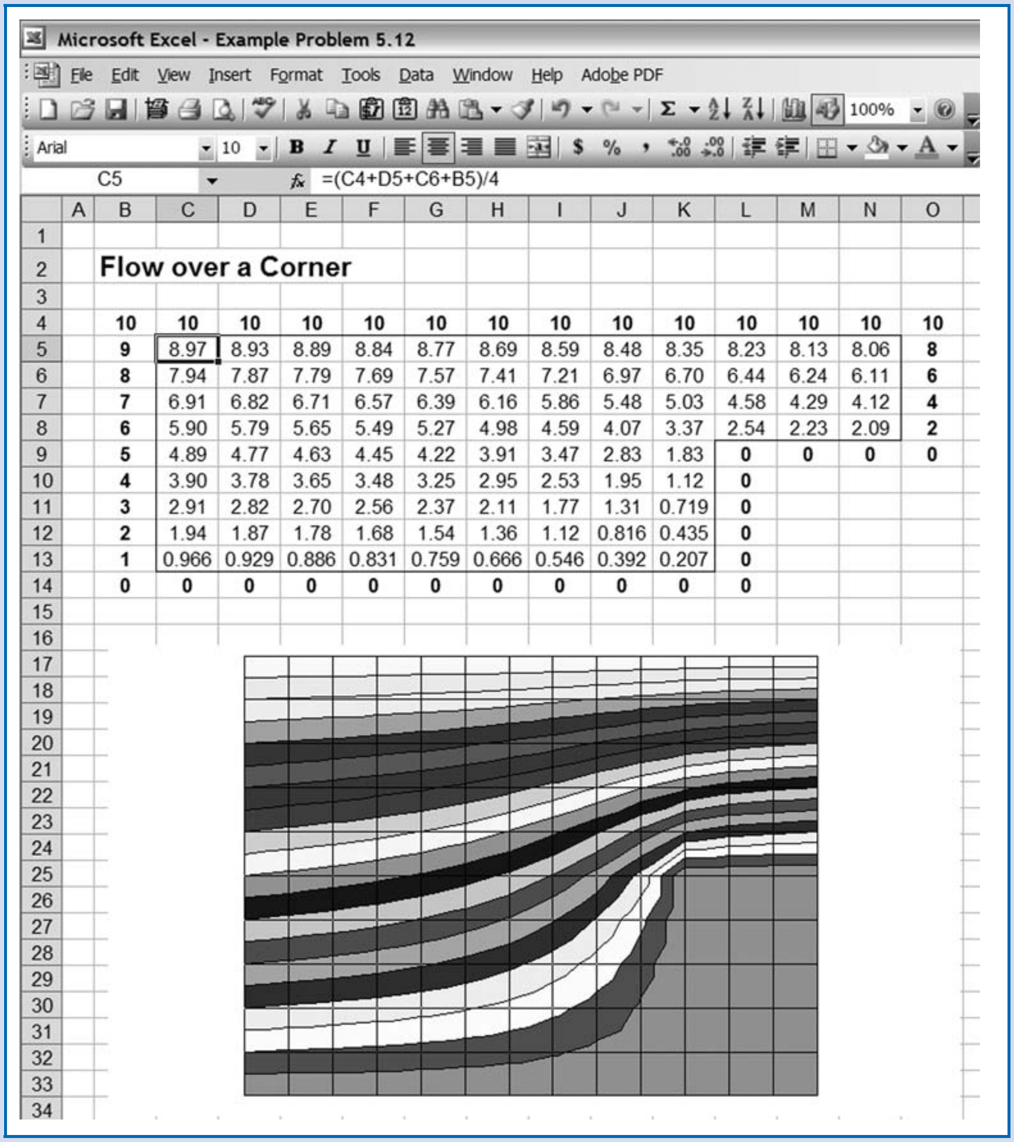
Solution: Use the numerical approximation of the Laplace equation.

Governing equation: $\psi_{i,j} = \dfrac{1}{4}\left(\psi_{i+1,j} + \psi_{i-1,j} + \psi_{i,j+1} + \psi_{i,j-1}\right)$

This is again convenient for solving using a spreadsheet such as *Excel*. Each cell in the spreadsheet represents a location in physical space, and the value in the cell represents the value of the stream function ψ at that location. Referring to the figure, we assign values of zero to a range of cells that represent the bottom of the channel. We then assign a value of 10 to a second range of cells to represent the top of the channel. (The choice of 10 is arbitrary for plotting purposes; all it determines is the speed values, not the streamline shapes.) Next, we assign a uniform distribution of values at the left and right ends, to generate uniform flow at those locations. All inserted values are shown in bold in the figure.

We can now enter formulas in the "interior" cells to compute the stream function. Instead of the above governing equation, it is more intuitive to rephrase it as

$$\psi = \frac{1}{4}(\psi_A + \psi_R + \psi_B + \psi_L)$$



	B	C	D	E	F	G	H	I	J	K	L	M	N	O
4	**10**	**10**	**10**	**10**	**10**	**10**	**10**	**10**	**10**	**10**	**10**	**10**	**10**	**10**
5	**9**	8.97	8.93	8.89	8.84	8.77	8.69	8.59	8.48	8.35	8.23	8.13	8.06	**8**
6	**8**	7.94	7.87	7.79	7.69	7.57	7.41	7.21	6.97	6.70	6.44	6.24	6.11	**6**
7	**7**	6.91	6.82	6.71	6.57	6.39	6.16	5.86	5.48	5.03	4.58	4.29	4.12	**4**
8	**6**	5.90	5.79	5.65	5.49	5.27	4.98	4.59	4.07	3.37	2.54	2.23	2.09	**2**
9	**5**	4.89	4.77	4.63	4.45	4.22	3.91	3.47	2.83	1.83	**0**	**0**	**0**	**0**
10	**4**	3.90	3.78	3.65	3.48	3.25	2.95	2.53	1.95	1.12	**0**			
11	**3**	2.91	2.82	2.70	2.56	2.37	2.11	1.77	1.31	0.719	**0**			
12	**2**	1.94	1.87	1.78	1.68	1.54	1.36	1.12	0.816	0.435	**0**			
13	**1**	0.966	0.929	0.886	0.831	0.759	0.666	0.546	0.392	0.207	**0**			
14	**0**	**0**	**0**	**0**	**0**	**0**	**0**	**0**	**0**	**0**	**0**			

where ψ_A, ψ_R, ψ_B, and ψ_L represent the values stored in the cells *A*bove, to the *R*ight, *B*elow, and to the *L*eft of the current cell. This formula is easy to enter— it is shown in cell C5 in the figure. Then it is copied into all interior cells, with one caveat: The spreadsheet will indicate an error of circular calculation. This is a warning that you appear to be

making an error; for example, cell C5 needs cell C6 to compute, but cell C6 needs cell C5! Recall that each interior cell value is the average of its neighbors. Circular math is usually not what we want, but in this case we do wish it to occur. We need to switch on *iteration* in the spreadsheet. In the case of *Excel*, it's under menu item *Tools/Options/Calculation*. Finally, we need to repeatedly iterate (in *Excel*, press the F9 key several times) until we have convergence; the values in the interior cells will repeatedly update until the variations in values is zero or trivial. After all this, the results can be plotted (using a surface plot), as shown.

We can see that the streamlines look much as we would anticipate, although in reality there would probably be flow separation at the corner. Note also a mathematical artifact in that there is slight oscillations of streamlines as they flow up the vertical surface; using a finer grid (by using many more cells) would reduce this.

This Example shows a simple numerical modeling of the Laplace equation. The *Excel* workbook for this problem can be modified for solving a variety of fluids problems that involve the Laplace equation.

Examples 5.11 and 5.12 provide guidance in using the PC to solve some simple CFD problems. We now turn to a somewhat more detailed description of some of the concepts behind CFD.

The Strategy of CFD

Broadly, the strategy of CFD is to replace the continuous problem domain with a discrete domain using a "grid" or "mesh." In the continuous domain, each flow variable is defined at every point in the domain. For instance, the pressure p in the continuous 1D domain shown in Fig. 5.15 would be given as

$$p = p(x), \qquad 0 \le x \le 1$$

In the discrete domain, each flow variable is defined only at the grid points. So, in the discrete domain in Fig. 5.15, the pressure would be defined only at the N grid points,

$$p_i = p(x_i), \quad i = 1, 2, \ldots, N$$

We can extend this continuous-to-discrete conversion to two or three dimensions. Figure 5.16 shows a 2D grid used for solving the flow over an airfoil. The grid points are the locations where the grid lines cross. In a CFD solution, we would directly solve for the relevant flow variables only at the grid points. The values at other locations are determined by interpolating the values at the grid points. The governing partial differential equations and boundary conditions are defined in terms of the continuous variables p, $\vec{V}$, and so on. We can approximate these in the discrete domain in terms of the discrete variables p_i, $\vec{V}_i$, and so on. Using this procedure, we end up with a discrete system that consists of a large set of coupled, algebraic equations in the discrete variables. Setting up the discrete system and solving it (which is a matrix inversion problem) involves a very large number of repetitive calculations, a task made possible only with the advent of modern computers.

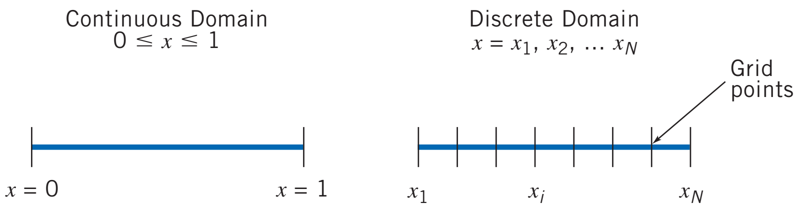


Fig. 5.15 Continuous and discrete domains for a one-dimensional problem.

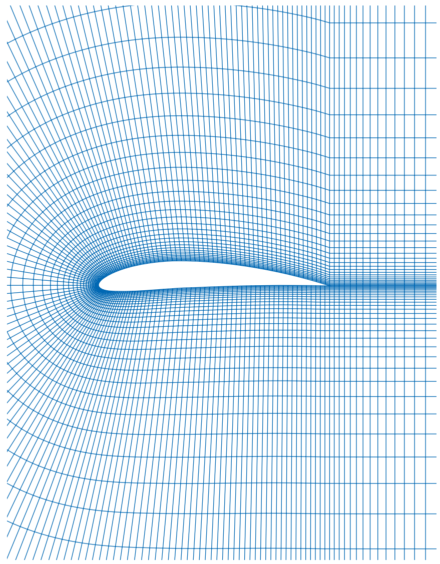


Fig. 5.16 Example of a grid used to solve for the flow around an airfoil.

Discretization Using the Finite-Difference Method

To keep the details simple, we will illustrate the process of going from the continuous domain to the discrete domain by applying it to the following simple 1D equation:

$$\frac{du}{dx} + u^m = 0; \qquad 0 \le x \le 1; \qquad u(0) = 1 \tag{5.31}$$

We'll first consider the case where $m = 1$, which is the case when the equation is linear. We'll later consider the nonlinear case $m = 2$. Keep in mind that the above problem is an initial-value problem, while the numerical solution procedure below is more suitable for boundary-value problems. Most CFD problems are boundary-value problems.

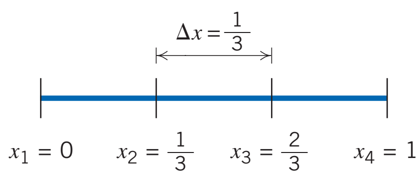


Fig. 5.17 A simple 1D grid with four grid points.

We'll derive a discrete representation of Eq. 5.31 with $m = 1$ on the rudimentary grid shown in Fig. 5.17. This grid has four equally spaced grid points, with $\Delta x = \frac{1}{3}$ being the spacing between successive points. Since the governing equation is valid at any grid point, we have

$$\left(\frac{du}{dx}\right)_i + u_i = 0 \tag{5.32}$$

where the subscript i represents the value at grid point x_i. In order to get an expression for $(du/dx)_i$ in terms of u values at the grid points, we expand u_{i-1} in a Taylor series:

$$u_{i-1} = u_i - \left(\frac{du}{dx}\right)_i \Delta x + \left(\frac{d^2u}{dx^2}\right)_i \frac{\Delta x^2}{2} - \left(\frac{d^3u}{dx^3}\right)_i \frac{\Delta x^3}{6} + \cdots$$

Rearranging this gives

$$\left(\frac{du}{dx}\right)_i = \frac{u_i - u_{i-1}}{\Delta x} + \left(\frac{d^2u}{dx^2}\right)_i \frac{\Delta x}{2} - \left(\frac{d^3u}{dx^3}\right)_i \frac{\Delta x^2}{6} + \cdots \tag{5.33}$$

We'll neglect the second-, third-, and higher-order terms on the right. Thus, the first term on the right is the finite-difference representation for $(du/dx)_i$ we are seeking. The error in $(du/dx)_i$ due to the neglected terms in the Taylor series is called the *truncation error*. In general, the truncation error is the difference between the differential equation and its finite-difference representation. The leading-order term in the truncation error in Eq. 5.33 is proportional to Δx. Equation 5.33 is rewritten as

$$\left(\frac{du}{dx}\right)_i = \frac{u_i - u_{i-1}}{\Delta x} + O(\Delta x) \tag{5.34}$$

where the last term is pronounced "order of delta x." The notation $O(\Delta x)$ has a precise mathematical meaning, which we will not go into here. Instead, in the interest of brevity, we'll return to it briefly later when we discuss the topic of grid convergence. Since the truncation error is proportional to the first power of Δx, this discrete representation is termed *first-order accurate*.

Using Eq. 5.34 in Eq. 5.32, we get the following discrete representation for our model equation:

$$\frac{u_i - u_{i-1}}{\Delta x} + u_i = 0 \tag{5.35}$$

Note that we have gone from a differential equation to an algebraic equation! Though we have not written it out explicitly, don't forget that the error in this representation is $O(\Delta x)$.

This method of deriving the discrete equation using Taylor's series expansions is called the *finite-difference method*. Keep in mind that most industrial CFD software packages use the *finite-volume* or *finite-element* discretization methods since they are better suited to modeling flow past complex geometries. We will stick with the finite-difference method in this text since it is the easiest to understand; the concepts discussed also apply to the other discretization methods.

Assembly of Discrete System and Application of Boundary Conditions

Rearranging the discrete equation, Eq. 5.35, we get

$$-u_{i-1} + (1 + \Delta x)u_i = 0$$

Applying this equation at grid points $i = 2, 3, 4$ for the 1D grid in Fig. 5.17 gives

$$-u_1 + (1 + \Delta x)u_2 = 0 \tag{5.36a}$$

$$-u_2 + (1 + \Delta x)u_3 = 0 \tag{5.36b}$$

$$-u_3 + (1 + \Delta x)u_4 = 0 \tag{5.36c}$$

The discrete equation cannot be applied at the left boundary ($i = 1$) since $u_{i-1} = u_0$ is not defined. Instead, we use the boundary condition to get

$$u_1 = 1 \tag{5.36d}$$

Equations 5.36 form a system of four simultaneous algebraic equations in the four unknowns u_1, u_2, u_3, and u_4. It's convenient to write this system in matrix form:

$$\begin{bmatrix} 1 & 0 & 0 & 0 \\ -1 & 1+\Delta x & 0 & 0 \\ 0 & -1 & 1+\Delta x & 0 \\ 0 & 0 & -1 & 1+\Delta x \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \begin{bmatrix} 1 \\ 0 \\ 0 \\ 0 \end{bmatrix} \tag{5.37}$$

In a general situation (e.g., 2D or 3D domains), we would apply the discrete equations to the grid points in the interior of the domain. For grid points at or near the boundary, we would apply a combination of the discrete equations and boundary conditions. In the end, one would obtain a system of simultaneous algebraic equations similar to Eqs. 5.36 and a matrix equation similar to Eq. 5.37, with the number of equations being equal to the number of independent discrete variables. The process is essentially the same as for the model equation above, with the details obviously being much more complex.

Solution of Discrete System

The discrete system (Eq. 5.37) for our own simple 1D example can be easily inverted, using any number of techniques of linear algebra, to obtain the unknowns at the grid points. For $\Delta x = \frac{1}{3}$, the solution is

$$u_1 = 1 \quad u_2 = \frac{3}{4} \quad u_3 = \frac{9}{16} \quad u_4 = \frac{27}{64}$$

The exact solution for Eq. 5.31 with $m = 1$ is easily shown to be

$$u_{\text{exact}} = e^{-x}$$

Figure 5.18 shows the comparison of the discrete solution obtained on the four-point grid with the exact solution, using *Excel*. The error is largest at the right boundary, where it is equal to 14.7 percent. [It also shows the results using eight points ($N = 8$, $\Delta x = \frac{1}{7}$) and sixteen points ($N = 16$, $\Delta x = \frac{1}{15}$), which we discuss below.]

In a practical CFD application, we would have thousands, even millions, of unknowns in the discrete system; if one were to use, say, a Gaussian elimination procedure to invert the calculations, it would be extremely time-consuming even with a fast computer. Hence a lot of work has gone into optimizing the matrix inversion in

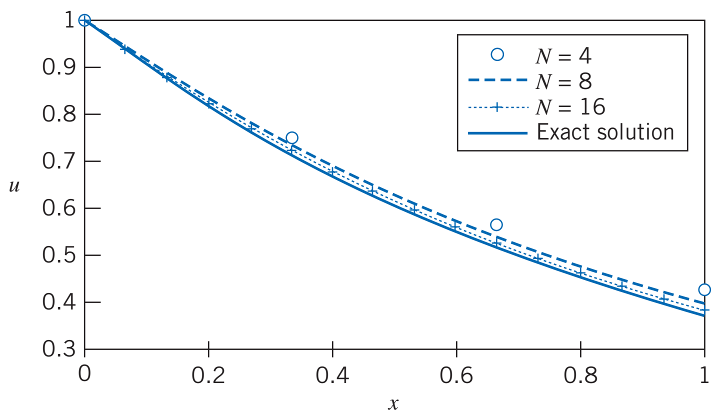


Fig. 5.18 Comparison of the numerical solution obtained on three different grids with the exact solution.

order to minimize the CPU time and memory required. The matrix to be inverted is sparse; that is, most of the entries in it are zeros. The nonzero entries are clustered around the diagonal since the discrete equation at a grid point contains only quantities at the neighboring grid points, as shown in Eq. 5.37. A CFD code would store only the nonzero values to minimize memory usage. It would also generally use an iterative procedure to invert the matrix; the longer one iterates, the closer one gets to the true solution for the matrix inversion. We'll return to this idea a little later.

Grid Convergence

While developing the finite-difference approximation for the 1D model problem (Eq. 5.37), we saw that the truncation error in our discrete system is $O(\Delta x)$. Hence we expect that as the number of grid points is increased and Δx is reduced, the error in the numerical solution would decrease and the agreement between the numerical and exact solutions would get better.

Let's consider the effect of increasing the number of grid points N on the numerical solution of the 1D problem. We'll consider $N = 8$ and $N = 16$ in addition to the $N = 4$ case solved previously. We repeat the above assembly and solution steps on each of these additional grids; instead of the 4×4 problem of Eq. 5.37, we end up with an 8×8 and a 16×16 problem, respectively. Figure 5.18 compares the results obtained (using *Excel*) on the three grids with the exact solution. As expected, the numerical error decreases as the number of grid points is increased (but this only goes so far—if we make Δx too small, we start to get round-off errors accumulating to make the results get worse!). When the numerical solutions obtained on different grids agree to within a level of tolerance specified by the user, they are referred to as "grid-converged" solutions. It is very important to investigate the effect of grid resolution on the solution in all CFD problems. We should never trust a CFD solution unless we are convinced that the solution is grid-converged to an acceptance level of tolerance (which will be problem dependent).

Let ε be some aggregate measure of the error in the numerical solution obtained on a specific grid. For the numerical solutions in Fig. 5.19, ε is, for instance, estimated as the RMS of the difference between the numerical and exact solutions:

$$\varepsilon = \sqrt{\frac{\sum_{i=1}^{N} (u_i - u_{i_{\text{exact}}})^2}{N}}$$

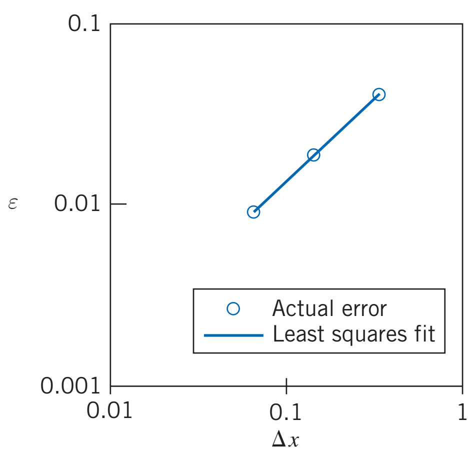


Fig. 5.19 The variation of the aggregate error ε with Δx.

It's reasonable to expect that

$$\varepsilon \propto \Delta x^n$$

Since the truncation error is $O(\Delta x)$ for our discretization scheme, we expect $n = 1$ (or more precisely, $n \to 1$ as $\Delta x \to 0$). The ε values for the three grids are plotted on a logarithmic scale in Fig. 5.19. The slope of the least squares fit gives the value of n. For Fig. 5.19, we get $n = 0.92$, which is quite close to 1. We expect that as the grid is refined further and Δx becomes progressively smaller, the value of n will approach 1. For a second-order scheme, we would expect $n \sim 2$; this means the discretization error will decrease twice as fast on refining the grid.

Dealing with Nonlinearity

The Navier–Stokes equations (Eqs. 5.27) contain nonlinear convection terms; for example, in Eq. 5.27a, the convective acceleration term, $u\partial u/\partial x + v\partial u/\partial y + w\partial u/\partial z$, has products of u with itself as well as with v and w. Phenomena such as turbulence and chemical reaction introduce additional nonlinearities. The highly nonlinear nature of the governing equations for a fluid makes it challenging to obtain accurate numerical solutions for complex flows of practical interest.

We will demonstrate the effect of nonlinearity by setting $m = 2$ in our simple 1D example, Eq. 5.31:

$$\frac{du}{dx} + u^2 = 0; \quad 0 \le x \le 1; \quad u(0) = 1$$

A first-order finite-difference approximation to this equation, analogous to that in Eq. 5.35 for $m = 1$, is

$$\frac{u_i - u_{i-1}}{\Delta x} + u_i^2 = 0 \tag{5.38}$$

This is a nonlinear algebraic equation with the u_i^2 term being the source of the nonlinearity.

The strategy that is adopted to deal with nonlinearity is to linearize the equations around a *guess value* of the solution and to iterate until the guess agrees with the solution to a specified tolerance level. We'll illustrate this on the above example. Let u_{g_i} be the guess for u_i. Define

$$\Delta u_i = u_i - u_{g_i}$$

Rearranging and squaring this equation gives

$$u_i^2 = u_{g_i}^2 + 2u_{g_i}\Delta u_i + (\Delta u_i)^2$$

Assuming that $\Delta u_i \ll u_{g_i}$, we can neglect the $(\Delta u_i)^2$ term to get

$$u_i^2 \approx u_{g_i}^2 + 2u_{g_i}\Delta u_i = u_{g_i}^2 + 2u_{g_i}(u_i - u_{g_i})$$

Thus

$$u_i^2 \approx 2u_{g_i}u_i - u_{g_i}^2 \tag{5.39}$$

The finite-difference approximation, Eq. 5.38, after linearization in u_i, becomes

$$\frac{u_i - u_{i-1}}{\Delta x} + 2u_{g_i}u_i - u_{g_i}^2 = 0 \tag{5.40}$$

Since the error due to linearization is $O(\Delta u^2)$, it tends to zero as $u_g \to u$.

In order to calculate the finite-difference approximation, Eq. 5.40, we need guess values u_g at the grid points. We start with an initial guess value in the first iteration. For each subsequent iteration, the u value obtained in the previous iteration is used as the guess value. We continue the iterations until they converge. We'll defer the discussion on how to evaluate convergence until a little later.

This is essentially the process used in CFD codes to linearize the nonlinear terms in the conservation equations, with the details varying depending on the code. The important points to remember are that the linearization is performed about a guess and that it is necessary to iterate through successive approximations until the iterations converge.

Direct and Iterative Solvers

We saw that we need to perform iterations to deal with the nonlinear terms in the governing equations. We next discuss another factor that makes it necessary to carry out iterations in practical CFD problems.

As an exercise, you can verify that the discrete equation system resulting from the finite-difference approximation of Eq. 5.40, on our four-point grid, is

$$\begin{bmatrix} 1 & 0 & 0 & 0 \\ -1 & 1+2\Delta x u_{g_2} & 0 & 0 \\ 0 & -1 & 1+2\Delta x u_{g_3} & 0 \\ 0 & 0 & -1 & 1+2\Delta x u_{g_4} \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \begin{bmatrix} 1 \\ \Delta x u_{g_2}^2 \\ \Delta x u_{g_3}^2 \\ \Delta x u_{g_4}^2 \end{bmatrix} \tag{5.41}$$

In a practical problem, one would usually have thousands to millions of grid points or cells so that each dimension of the above matrix would be of the order of a million (with most of the elements being zeros). Inverting such a matrix directly would take a prohibitively large amount of memory, so instead the matrix is inverted using an iterative scheme as discussed below.

Rearrange the finite-difference approximation, Eq. 5.40, at grid point i so that u_i is expressed in terms of the values at the neighboring grid points and the guess values:

$$u_i = \frac{u_{i-1} + \Delta x\, u_{g_i}^2}{1 + 2\Delta x\, u_{g_i}}$$

If a neighboring value at the current iteration level is not available, we use the guess value for it. Let's say that we sweep from right to left on our grid; that is, we update u_4, then u_3, and finally u_2 in each iteration. In any iteration, u_{i-1} is not available while updating u_i and so we use the guess value $u_{g_{i-1}}$ for it instead:

$$u_i = \frac{u_{g_{i-1}} + \Delta x\, u_{g_i}^2}{1 + 2\Delta x\, u_{g_i}} \tag{5.42}$$

Since we are using the guess values at neighboring points, we are effectively obtaining only an approximate solution for the matrix inversion in Eq. 5.41 during each iteration, but in the process we have greatly reduced the memory required for the inversion. This trade-off is a good strategy since it doesn't make sense to expend a great deal of resources to do an exact matrix inversion when the matrix elements depend on guess values that are continuously being refined. We have in effect combined the iteration to handle nonlinear terms with the iteration for matrix inversion into a single iteration process. Most importantly, as the iterations converge and $u_g \to u$, the approximate solution for the matrix inversion tends towards the exact solution for the inversion, since the error introduced by using u_g instead of u in Eq. 5.42 also tends

to zero. We arrive at the solution without explicitly forming the matrix system (Eq. 5.41), which greatly simplifies the computer implementation.

Thus, iteration serves two purposes:

1. It allows for efficient matrix inversion with greatly reduced memory requirements.
2. It enables us to solve nonlinear equations.

In steady problems, a common and effective strategy used in CFD codes is to solve the unsteady form of the governing equations and "march" the solution in time until the solution converges to a steady value. In this case, each time step is effectively an iteration, with the guess value at any time level being given by the solution at the previous time level.

Iterative Convergence

Recall that as $u_g \to u$, the linearization and matrix inversion errors tend to zero. Hence we continue the iteration process until some selected measure of the difference between u_g and u, referred to as the residual, is "small enough." We could, for instance, define the residual R as the RMS value of the difference between u and u_g on the grid:

$$R \equiv \sqrt{\frac{\sum_{i=1}^{N} (u_i - u_{g_i})^2}{N}}$$

It's useful to scale this residual with the average value of u in the domain. Scaling ensures that the residual is a *relative* rather than an *absolute* measure. Scaling the above residual by dividing by the average value of u gives

$$R = \left(\sqrt{\frac{\sum_{i=1}^{N} (u_i - u_{g_i})^2}{N}}\right)\left(\frac{N}{\sum_{i=1}^{N} u_i}\right) = \frac{\sqrt{N \sum_{i=1}^{N} (u_i - u_{g_i})^2}}{\sum_{i=1}^{N} u_i} \tag{5.43}$$

In our nonlinear 1D example, we'll take the initial guess at all grid points to be equal to the value at the left boundary, that is, $u_g^{(1)} = 1$ (where $^{(1)}$ signifies the first iteration). In each iteration, we update u_g, sweep from right to left on the grid updating, in turn, u_4, u_3, and u_2 using Eq. 5.42, and calculate the residual using Eq. 5.43. We'll terminate the iterations when the residual falls below 10^{-9} (this is referred to as the *convergence criterion*). The variation of the residual with iterations is shown in Fig. 5.20. Note that a logarithmic scale is used for the ordinate. The iterative process converges to a level smaller than 10^{-9} in only six iterations. In more complex problems, many more iterations would be necessary for achieving convergence.

The solution after two, four, and six iterations and the exact solution are shown in Fig. 5.21. It can easily be verified that the exact solution is given by

$$u_{\text{exact}} = \frac{1}{x+1}$$

The solutions for four and six iterations are indistinguishable on the graph. This is another indication that the solution has converged. The converged solution doesn't agree well with the exact solution because we are using a coarse grid for which the truncation error is relatively large (we will repeat this problem with finer grids as problems at the end of the chapter). The iterative convergence error, which is of order 10^{-9}, is swamped by the truncation error, which is of order 10^{-1}. So driving the residual down to 10^{-9} when the truncation error is of order 10^{-1} is obviously a waste

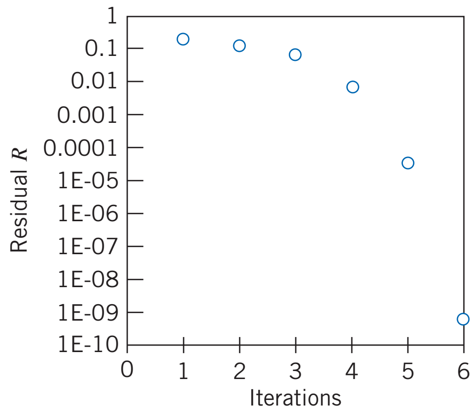


Fig. 5.20 Convergence history for the model nonlinear problem.

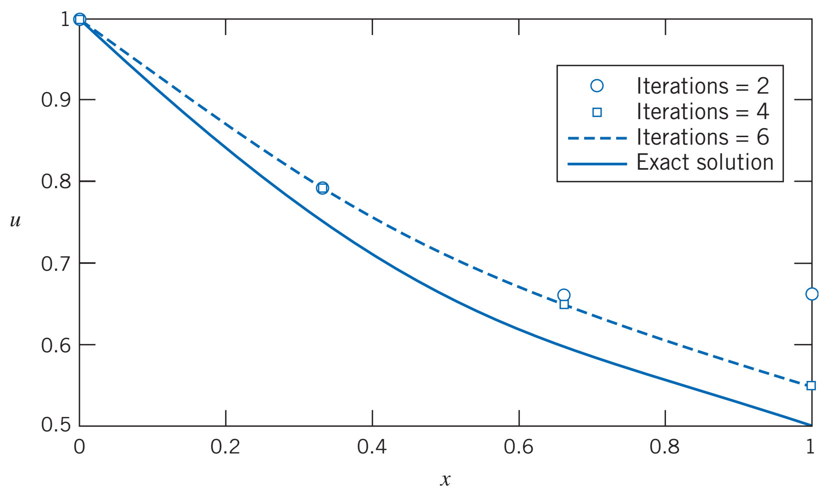


Fig. 5.21 Progression of the iterative solution.

of computing resources. In an efficient calculation, both errors would be set at comparable levels, and less than a tolerance level that was chosen by the user. The agreement between the numerical and exact solutions should get much better on refining the grid, as was the linear case (for $m = 1$). Different CFD codes use slightly different definitions for the residual. You should always read the documentation from the application to understand how the residual is calculated.

Concluding Remarks

In this section we have introduced some simple ways of using a spreadsheet for the numerical solution of two types of fluid mechanics problems. Examples 5.11 and 5.12 show how certain 1D and 2D flows may be computed. We then studied some concepts in more detail, such as convergence criteria, involved with numerical methods and CFD, by considering a first-order ODE. In our simple 1D example, the iterations converged very rapidly. In practice, one encounters many instances when the iterative process doesn't converge or converges lethargically. Hence, it's useful to know *a priori* the conditions under which a given numerical scheme converges. This is determined by performing a stability analysis of the numerical scheme. Stability analysis of numerical schemes and the various stabilization strategies used to overcome nonconvergence are very important topics and necessary for you to explore if you decide to delve further into the topic of CFD.

VIDEO

Fully Turbulent Duct Flow.

Many engineering flows are turbulent, characterized by large, nearly random fluctuations in velocity and pressure in both space and time. Turbulent flows often occur in the limit of high Reynolds numbers. For most turbulent flows, it is not possible to resolve the vast range of time and length scales, even with powerful computers. Instead, one solves for a statistical average of the flow properties. In order to do this, it is necessary to augment the governing equations with a turbulence model. Unfortunately, there is no single turbulence model that is uniformly valid for all flows, so most CFD packages allow you to choose from among several models. Before you use a turbulence model, you need to understand its possibilities and limitations for the type of flow being considered.

In this brief introduction we have tried to explain some of the concepts behind CFD. Because it is so difficult and time consuming to develop CFD code, most engineers use commercial packages such as *Fluent* [6] and *STAR-CD* [7]. This introduction will have hopefully indicated for you the complexity behind those applications, so that they are not completely a "black box" of magic tricks.

5.6 Summary and Useful Equations

In this chapter we have:

- ✓ Derived the differential form of the conservation of mass (continuity) equation in vector form as well as in rectangular and cylindrical coordinates.
- ✓ *Defined the stream function ψ for a two-dimensional incompressible flow and learned how to derive the velocity components from it, as well as to find ψ from the velocity field.
- ✓ Learned how to obtain the total, local, and convective accelerations of a fluid particle from the velocity field.
- ✓ Presented examples of fluid particle translation and rotation, and both linear and angular deformation.
- ✓ Defined vorticity and circulation of a flow.
- ✓ Derived, and solved for simple cases, the Navier–Stokes equations, and discussed the physical meaning of each term.
- ✓ *Been introduced to some basis ideas behind computational fluid dynamics.

We have also explored such ideas as how to determine whether a flow is incompressible by using the velocity field and, given one velocity component of a two-dimensional incompressible flow field, how to derive the other velocity component.

In this chapter we studied the effects of viscous stresses on fluid particle deformation and rotation; in the next chapter we examine flows for which viscous effects are negligible.

Note: Most of the Useful Equations in the table below have a number of constraints or limitations—*be sure to refer to their page numbers for details*!

Useful Equations

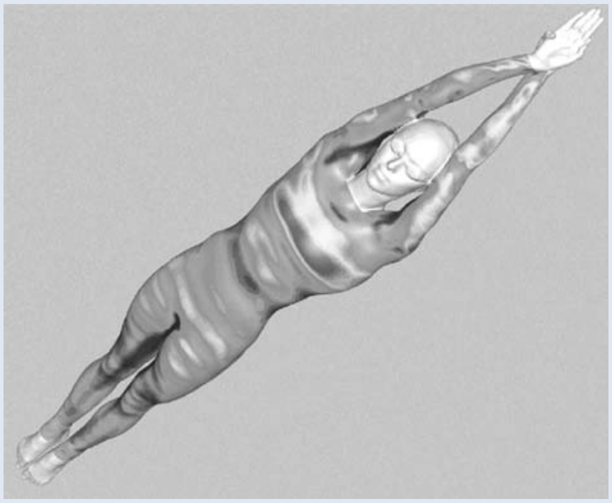
Continuity equation (general, rectangular coordinates):	$\dfrac{\partial \rho u}{\partial x} + \dfrac{\partial \rho v}{\partial y} + \dfrac{\partial \rho w}{\partial z} + \dfrac{\partial \rho}{\partial t} = 0$ $\nabla \cdot \rho \vec{V} + \dfrac{\partial \rho}{\partial t} = 0$	**(5.1a)** **(5.1b)**	Page 175
Continuity equation (incompressible, rectangular coordinates):	$\dfrac{\partial u}{\partial x} + \dfrac{\partial v}{\partial y} + \dfrac{\partial w}{\partial z} = \nabla \cdot \vec{V} = 0$	**(5.1c)**	Page 175
Continuity equation (steady, rectangular coordinates):	$\dfrac{\partial \rho u}{\partial x} + \dfrac{\partial \rho v}{\partial y} + \dfrac{\partial \rho w}{\partial z} = \nabla \cdot \rho \vec{V} = 0$	**(5.1d)**	Page 175
Continuity equation (general, cylindrical coordinates):	$\dfrac{1}{r}\dfrac{\partial (r \rho V_r)}{\partial r} + \dfrac{1}{r}\dfrac{\partial (\rho V_\theta)}{\partial \theta} + \dfrac{\partial (\rho V_z)}{\partial z} + \dfrac{\partial \rho}{\partial t} = 0$ $\nabla \cdot \rho \vec{V} + \dfrac{\partial \rho}{\partial t} = 0$	**(5.2a)** **(5.1b)**	Pages 178

*This section may be omitted without loss of continuity in the text material.

Continuity equation (incompressible, cylindrical coordinates):	$\frac{1}{r}\frac{\partial(rV_r)}{\partial r}+\frac{1}{r}\frac{\partial V_\theta}{\partial\theta}+\frac{\partial V_z}{\partial z}=\nabla\cdot\vec{V}=0$	**(5.2b)**	Page 178
Continuity equation (steady, cylindrical coordinates):	$\frac{1}{r}\frac{\partial(r\rho V_r)}{\partial r}+\frac{1}{r}\frac{\partial(\rho V_\theta)}{\partial\theta}+\frac{\partial(\rho V_z)}{\partial z}=\nabla\cdot\rho\vec{V}=0$	**(5.2c)**	Page 180
Continuity equation (2D, incompressible, rectangular coordinates):	$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$	**(5.3)**	Page 181
Stream function (2D, incompressible, rectangular coordinates):	$u\equiv\frac{\partial\psi}{\partial y}$ and $v\equiv-\frac{\partial\psi}{\partial x}$	**(5.4)**	Page 181
Continuity equation (2D, incompressible, cylindrical coordinates):	$\frac{\partial(rV_r)}{\partial r}+\frac{\partial V_\theta}{\partial\theta}=0$	**(5.7)**	Page 182
Stream function (2D, incompressible, cylindrical coordinates):	$V_r\equiv\frac{1}{r}\frac{\partial\psi}{\partial\theta}$ and $V_\theta\equiv-\frac{\partial\psi}{\partial r}$	**(5.8)**	Page 183
Particle acceleration (rectangular coordinates):	$\frac{D\vec{V}}{Dt}\equiv\vec{a}_p=u\frac{\partial\vec{V}}{\partial x}+v\frac{\partial\vec{V}}{\partial y}+w\frac{\partial\vec{V}}{\partial z}+\frac{\partial\vec{V}}{\partial t}$	**(5.9)**	Page 186
Particle acceleration components in rectangular coordinates:	$a_{x_p}=\frac{Du}{Dt}=u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}+\frac{\partial u}{\partial t}$	**(5.11a)**	Page 187
	$a_{y_p}=\frac{Dv}{Dt}=u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}+\frac{\partial v}{\partial t}$	**(5.11b)**	
	$a_{z_p}=\frac{Dw}{Dt}=u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}+\frac{\partial w}{\partial t}$	**(5.11c)**	
Particle acceleration components in cylindrical coordinates:	$a_{r_p}=V_r\frac{\partial V_r}{\partial r}+\frac{V_\theta}{r}\frac{\partial V_r}{\partial\theta}-\frac{V_\theta^2}{r}+V_z\frac{\partial V_r}{\partial z}+\frac{\partial V_r}{\partial t}$	**(5.12a)**	Page 188
	$a_{\theta_p}=V_r\frac{\partial V_\theta}{\partial r}+\frac{V_\theta}{r}\frac{\partial V_\theta}{\partial\theta}+\frac{V_rV_\theta}{r}+V_z\frac{\partial V_\theta}{\partial z}+\frac{\partial V_\theta}{\partial t}$	**(5.12b)**	
	$a_{z_p}=V_r\frac{\partial V_z}{\partial r}+\frac{V_\theta}{r}\frac{\partial V_z}{\partial\theta}+V_z\frac{\partial V_z}{\partial z}+\frac{\partial V_z}{\partial t}$	**(5.12c)**	
Navier–Stokes equations (incompressible, constant viscosity):	$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=\rho g_x-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2u}{\partial x^2}+\frac{\partial^2u}{\partial y^2}+\frac{\partial^2u}{\partial z^2}\right)$	**(5.27a)**	Page 200, 201
	$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=\rho g_y-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2v}{\partial x^2}+\frac{\partial^2v}{\partial y^2}+\frac{\partial^2v}{\partial z^2}\right)$	**(5.27b)**	
	$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=\rho g_z-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2w}{\partial x^2}+\frac{\partial^2w}{\partial y^2}+\frac{\partial^2w}{\partial z^2}\right)$	**(5.27c)**	

Case Study

Olympic Swimming and Bobsledding



CFD simulation of water flow over typical female elite swimmer in the glide position showing contours of shear stress. (Courtesy of Speedo and Fluent Inc.)

Athletes in many competitive sports are using technology to gain an advantage. In recent years, Fastskin® fabric has been developed by Speedo. This material allows the lowest-drag racing swimwear in the world to be developed. The fabric mimics the rough denticles of sharks' skin to reduce drag in key areas of the body. (Shark scales are tiny compared with those of most fishes and have a toothlike structure, called dermal denticles—literally, "tiny skin teeth." These denticles are nature's way of reducing drag on the shark.) Detailed design of swimsuits was based on tests in a water flume and on computational fluid dynamics (CFD) analyses. The figure shows an example of the results obtained. To optimize the suits, the results were used to guide the position of the seams; gripper panels on the underside of the forearms; and "vortex" riblets on the chest, shoulders, and back of the suit—as well as the positioning of different patches of fabric and fabric coatings.

The same technology is now being used to make outfits for athletes in the bobsled and luge events in the winter Olympics. The fabric has been modified, based on wind tunnel tests, to reduce drag based on the airflow direction unique to sledding sports. The new outfits also eliminate most of the fabric vibration (a major source of drag) found in other speed suits.

For both summer and winter sports, the ability to perform experimental and theoretical fluid dynamics analysis and make design changes based on these can make the difference in speed of several percent—the difference between silver and gold!

References

1. Li, W. H., and S. H. Lam, *Principles of Fluid Mechanics*. Reading, MA: Addison-Wesley, 1964.
2. Daily, J. W., and D. R. F. Harleman, *Fluid Dynamics*. Reading, MA: Addison-Wesley, 1966.
3. Schlichting, H., *Boundary-Layer Theory*, 7th ed. New York: McGraw-Hill, 1979.
4. White, F. M., *Viscous Fluid Flow*, 3rd ed. New York: McGraw-Hill, 2000.
5. Sabersky, R. H., A. J. Acosta, E. G. Hauptmann, and E. M. Gates, *Fluid Flow—A First Course in Fluid Mechanics*, 4th ed. New Jersey: Prentice Hall, 1999.
6. *Fluent*. Fluent Incorporated, Centerra Resources Park, 10 Cavendish Court, Lebanon, NH 03766 (www.fluent.com).
7. *STAR-CD*. Adapco, 60 Broadhollow Road, Melville, NY 11747 (www.cd-adapco.com).
8. Chapra, S. C., and R. P. Canale, *Numerical Methods for Engineers*, 5th ed. New York: McGraw-Hill, 2005.
9. Epperson, J. F., *An Introduction to Numerical Methods and Analysis*, rev. ed. New York: Wiley, 2007.

Problems

Conservation of Mass

5.1 Which of the following sets of equations represent possible two-dimensional incompressible flow cases?
(a) $u = 2x^2 + y^2 - x^2y$; $v = x^3 + x(y^2 - 4y)$
(b) $u = 2xy - x^2y$; $v = 2xy - y^2 + x^2$
(c) $u = x^2t + 2y$; $v = xt^2 - yt$
(d) $u = (2x + 4y)xt$; $v = -3(x + y)yt$

5.2 Which of the following sets of equations represent possible three-dimensional incompressible flow cases?
(a) $u = 2y^2 + 2xz$; $v = -2yz + 6x^2yz$; $w = 3x^2z^2 + x^3y^4$
(b) $u = xyzt$; $v = -xyzt^2$; $w = z^2(xt^2 - yt)$
(c) $u = x^2 + 2y + z^2$; $v = x - 2y + z$; $w = -2xz + y^2 + 2z$

5.3 For a flow in the xy plane, the x component of velocity is given by $u = Ax(y - B)$, where $A = 1\ \text{ft}^{-1}\cdot\text{s}^{-1}$, $B = 6$ ft, and x

and y are measured in feet. Find a possible y component for steady, incompressible flow. Is it also valid for unsteady, incompressible flow? Why? How many y components are possible?

5.4 The three components of velocity in a velocity field are given by $u = Ax + By + Cz$, $v = Dx + Ey + Fz$, and $w = Gx + Hy + Jz$. Determine the relationship among the coefficients A through J that is necessary if this is to be a possible incompressible flow field.

5.5 For a flow in the xy plane, the x component of velocity is given by $u = 3x^2y - y^3$. Determine a possible y component for steady, incompressible flow. Is it also valid for unsteady, incompressible flow? Why? How many possible y components are there?

5.6 The x component of velocity in a steady, incompressible flow field in the xy plane is $u = A/x$, where $A = 2\ \text{m}^2/\text{s}$, and x is measured in meters. Find the simplest y component of velocity for this flow field.

5.7 The y component of velocity in a steady, incompressible flow field in the xy plane is $v = Axy(x^2 - y^2)$, where $A = 3\ m^{-3}{\cdot}\text{s}^{-1}$ and x and y are measured in meters. Find the simplest x component of velocity for this flow field.

5.8 The y component of velocity in a steady incompressible flow field in the xy plane is

$$v = \frac{2xy}{(x^2 + y^2)^2}$$

Show that the simplest expression for the x component of velocity is

$$u = \frac{1}{(x^2 + y^2)} - \frac{2y^2}{(x^2 + y^2)^2}$$

5.9 The x component of velocity in a steady incompressible flow field in the xy plane is $u = Ae^{x/b}\cos(y/b)$, where $A = 10$ m/s, $b = 5$ m, and x and y are measured in meters. Find the simplest y component of velocity for this flow field.

5.10 A crude approximation for the x component of velocity in an incompressible laminar boundary layer is a linear variation from $u = 0$ at the surface ($y = 0$) to the freestream velocity, U, at the boundary-layer edge ($y = \delta$). The equation for the profile is $u = Uy/\delta$, where $\delta = cx^{1/2}$ and c is a constant. Show that the simplest expression for the y component of velocity is $v = uy/4x$. Evaluate the maximum value of the ratio v/U, at a location where $x = 0.5$ m and $\delta = 5$ mm.

5.11 A useful approximation for the x component of velocity in an incompressible laminar boundary layer is a parabolic variation from $u = 0$ at the surface ($y = 0$) to the freestream velocity, U, at the edge of the boundary layer ($y = \delta$). The equation for the profile is $u/U = 2(y/\delta) - (y/\delta)^2$, where $\delta = cx^{1/2}$ and c is a constant. Show that the simplest expression for the y component of velocity is

$$\frac{v}{U} = \frac{\delta}{x}\left[\frac{1}{2}\left(\frac{y}{\delta}\right)^2 - \frac{1}{3}\left(\frac{y}{\delta}\right)^3\right]$$

Plot v/U versus y/δ to find the location of the maximum value of the ratio v/U. Evaluate the ratio where $\delta = 5$ mm and $x = 0.5$ m.

5.12 A useful approximation for the x component of velocity in an incompressible laminar boundary layer is a sinusoidal variation from $u = 0$ at the surface ($y = 0$) to the freestream velocity, U, at the edge of the boundary layer ($y = \delta$). The equation for the profile is $u = U\sin(\pi y/2\delta)$, where $\delta = cx^{1/2}$ and c is a constant. Show that the simplest expression for the y component of velocity is

$$\frac{v}{U} = \frac{1}{\pi}\frac{\delta}{x}\left[\cos\left(\frac{\pi}{2}\frac{y}{\delta}\right) + \left(\frac{\pi}{2}\frac{y}{\delta}\right)\sin\left(\frac{\pi}{2}\frac{y}{\delta}\right) - 1\right]$$

Plot u/U and v/U versus y/δ, and find the location of the maximum value of the ratio v/U. Evaluate the ratio where $x = 0.5$ m and $\delta = 5$ mm.

5.13 A useful approximation for the x component of velocity in an incompressible laminar boundary layer is a cubic variation from $u = 0$ at the surface ($y = 0$) to the freestream velocity, U, at the edge of the boundary layer ($y = \delta$). The equation for the profile is $u/U = \frac{3}{2}(y/\delta) - \frac{1}{2}(y/\delta)^3$, where $\delta = cx^{1/2}$ and c is a constant. Derive the simplest expression for v/U, the y component of velocity ratio. Plot u/U and v/U versus y/δ, and find the location of the maximum value of the ratio v/U. Evaluate the ratio where $\delta = 5$ mm and $x = 0.5$ m.

5.14 For a flow in the xy plane, the x component of velocity is given by $u = Ax^2y^2$, where $A = 0.3\ \text{m}^{-3}\cdot\text{s}^{-1}$, and x and y are measured in meters. Find a possible y component for steady, incompressible flow. Is it also valid for unsteady, incompressible flow? Why? How many possible y components are there? Determine the equation of the streamline for the simplest y component of velocity. Plot the streamlines through points (1, 4) and (2, 4).

5.15 The y component of velocity in a steady, incompressible flow field in the xy plane is $v = -Bxy^3$, where $B = 0.2\ \text{m}^{-3}\cdot\text{s}^{-1}$, and x and y are measured in meters. Find the simplest x component of velocity for this flow field. Find the equation of the streamlines for this flow. Plot the streamlines through points (1, 4) and (2, 4).

5.16 Consider a water stream from a jet of an oscillating lawn sprinkler. Describe the corresponding pathline and streakline.

5.17 Derive the differential form of conservation of mass in rectangular coordinates by expanding the *products* of density and the velocity components, ρu, ρv, and ρw, in a Taylor series about a point O. Show that the result is identical to Eq. 5.1a.

5.18 Which of the following sets of equations represent possible incompressible flow cases?

(a) $V_r = U\cos\theta$; $V_\theta = -U\sin\theta$
(b) $V_r = -q/2\pi r$; $V_\theta = K/2\pi r$
(c) $V_r = U\cos\theta\,[1 - (a/r)^2]$; $V_\theta = -U\sin\theta[1 + (a/r)^2]$

5.19 Which of the following sets of equations represent(s) possible incompressible flow cases?

(a) $V_r = -K/r$; $V_\theta = 0$
(b) $V_r = 0$; $V_\theta = K/r$
(c) $V_r = -K\cos\theta/r^2$; $V_\theta = -K\sin\theta/r^2$

5.20 For an incompressible flow in the $r\theta$ plane, the r component of velocity is given as $V_r = U\cos\theta$.

(a) Determine a possible θ component of velocity.
(b) How many possible θ components are there?

5.21 For an incompressible flow in the $r\theta$ plane, the r component of velocity is given as $V_r = -\Lambda \cos\theta/r^2$. Determine a possible θ component of velocity. How many possible θ components are there?

5.22 A viscous liquid is sheared between two parallel disks of radius R, one of which rotates while the other is fixed. The velocity field is purely tangential, and the velocity varies linearly with z from $V_\theta = 0$ at $z = 0$ (the fixed disk) to the velocity of the rotating disk at its surface ($z = h$). Derive an expression for the velocity field between the disks.

5.23 Evaluate $\nabla \cdot \rho\vec{V}$ in cylindrical coordinates. Use the definition of ∇ in cylindrical coordinates. Substitute the velocity vector and perform the indicated operations, using the hint in footnote 1 on page 178. Collect terms and simplify; show that the result is identical to Eq. 5.2c.

Stream Function for Two-Dimensional Incompressible Flow

***5.24** A velocity field in cylindrical coordinates is given as $\vec{V} = \hat{e}_r A/r + \hat{e}_\theta B/r$, where A and B are constants with dimensions of m^2/s. Does this represent a possible incompressible flow? Sketch the streamline that passes through the point $r_0 = 1$ m, $\theta = 90°$ if $A = B = 1$ m^2/s, if $A = 1$ m^2/s and $B = 0$, and if $B = 1$ m^2/s and $A = 0$.

***5.25** The velocity field for the viscometric flow of Example 5.7 is $\vec{V} = U(y/h)\hat{i}$. Find the stream function for this flow. Locate the streamline that divides the total flow rate into two equal parts.

***5.26** Determine the family of stream functions ψ that will yield the velocity field $\vec{V} = 2y(2x+1)\hat{i} + [x(x+1) - 2y^2]\hat{j}$.

***5.27** Does the velocity field of Problem 5.24 represent a possible incompressible flow case? If so, evaluate and sketch the stream function for the flow. If not, evaluate the rate of change of density in the flow field.

***5.28** The stream function for a certain incompressible flow field is given by the expression $\psi = -Ur\sin\theta + q\theta/2\pi$. Obtain an expression for the velocity field. Find the stagnation point(s) where $|\vec{V}| = 0$, and show that $\psi = 0$ there.

***5.29** Consider a flow with velocity components $u = z(3x^2 - z^2)$, $v = 0$, and $w = x(x^2 - 3z^2)$.
(a) Is this a one-, two-, or three-dimensional flow?
(b) Demonstrate whether this is an incompressible flow.
(c) If possible, derive a stream function for this flow.

***5.30** An incompressible frictionless flow field is specified by the stream function $\psi = -5Ax - 2Ay$, where $A = 2$ m/s, and x and y are coordinates in meters.
(a) Sketch the streamlines $\psi = 0$ and $\psi = 5$, and indicate the direction of the velocity vector at the point (0, 0) on the sketch.
(b) Determine the magnitude of the flow rate between the streamlines passing through (2, 2) and (4, 1).

***5.31** A linear velocity profile was used to model flow in a laminar incompressible boundary layer in Problem 5.10. Derive the stream function for this flow field. Locate streamlines at one-quarter and one-half the total volume flow rate in the boundary layer.

***5.32** A parabolic velocity profile was used to model flow in a laminar incompressible boundary layer in Problem 5.11. Derive the stream function for this flow field. Locate streamlines at one-quarter and one-half the total volume flow rate in the boundary layer.

***5.33** Derive the stream function that represents the sinusoidal approximation used to model the x component of velocity for the boundary layer of Problem 5.12. Locate streamlines at one-quarter and one-half the total volume flow rate in the boundary layer.

***5.34** A cubic velocity profile was used to model flow in a laminar incompressible boundary layer in Problem 5.13. Derive the stream function for this flow field. Locate streamlines at one-quarter and one-half the total volume flow rate in the boundary layer.

***5.35** A rigid-body motion was modeled in Example 5.6 by the velocity field $\vec{V} = r\omega\hat{e}_\theta$. Find the stream function for this flow. Evaluate the volume flow rate per unit depth between $r_1 = 0.10$ m and $r_2 = 0.12$ m, if $\omega = 0.5$ rad/s. Sketch the velocity profile along a line of constant θ. Check the flow rate calculated from the stream function by integrating the velocity profile along this line.

***5.36** In a parallel one-dimensional flow in the positive x direction, the velocity varies linearly from zero at $y = 0$ to 30 m/s at $y = 1.5$ m. Determine an expression for the stream function, ψ. Also determine the y coordinate above which the volume flow rate is half the total between $y = 0$ and $y = 1.5$ m.

***5.37** Example 5.6 showed that the velocity field for a free vortex in the $r\theta$ plane is $\vec{V} = \hat{e}_\theta C/r$. Find the stream function for this flow. Evaluate the volume flow rate per unit depth between $r_1 = 0.20$ m and $r_2 = 0.24$ m, if $C = 0.3$ m^2/s. Sketch the velocity profile along a line of constant θ. Check the flow rate calculated from the stream function by integrating the velocity profile along this line.

Motion of a Fluid Particle (Kinematics)

5.38 Consider the flow field given by $\vec{V} = xy^2\hat{i} - \frac{1}{3}y^3\hat{j} + xy\hat{k}$. Determine (a) the number of dimensions of the flow, (b) if it is a possible incompressible flow, and (c) the acceleration of a fluid particle at point $(x, y, z) = (1, 2, 3)$.

5.39 Consider the velocity field $\vec{V} = A(x^4 - 6x^2y^2 + y^4)\hat{i} + A(4xy^3 - 4x^3y)\hat{j}$ in the xy plane, where $A = 0.25$ m$^{-3}\cdot$s^{-1}, and the coordinates are measured in meters. Is this a possible incompressible flow field? Calculate the acceleration of a fluid particle at point $(x, y) = (2, 1)$.

5.40 Consider the flow field given by $\vec{V} = ax^2y\hat{i} - by\hat{j} + cz^2\hat{k}$, where $a = 2$ m$^{-2}\cdot$s^{-1}, $b = 2$ s^{-1}, and $c = 1$ m$^{-1}\cdot$s^{-1}. Determine (a) the number of dimensions of the flow, (b) if it is a possible incompressible flow, and (c) the acceleration of a fluid particle at point $(x, y, z) = (2, 1, 3)$.

5.41 The x component of velocity in a steady, incompressible flow field in the xy plane is $u = A(x^5 - 10x^3y^2 + 5xy^4)$, where

*These problems require material from sections that may be omitted without loss of continuity in the text material.

$A = 2\ \text{m}^{-4} \cdot \text{s}^{-1}$ and x is measured in meters. Find the simplest y component of velocity for this flow field. Evaluate the acceleration of a fluid particle at point $(x, y) = (1, 3)$.

5.42 The velocity field within a laminar boundary layer is approximated by the expression

$$\vec{V} = \frac{AUy}{x^{1/2}}\hat{i} + \frac{AUy^2}{4x^{3/2}}\hat{j}$$

In this expression, $A = 141\ \text{m}^{-1/2}$, and $U = 0.240$ m/s is the freestream velocity. Show that this velocity field represents a possible incompressible flow. Calculate the acceleration of a fluid particle at point $(x, y) = (0.5\ \text{m}, 5\ \text{mm})$. Determine the slope of the streamline through the point.

5.43 Wave flow of an incompressible fluid into a solid surface follows a sinusoidal pattern. Flow is two-dimensional with the x axis normal to the surface and y axis along the wall. The x component of the flow follows the pattern

$$u = Ax \sin\left(\frac{2\pi t}{T}\right)$$

Determine the y component of flow (v) and the convective and local components of the acceleration vector.

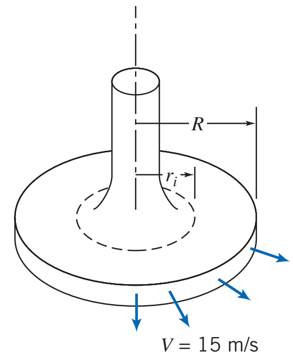
5.44 The y component of velocity in a two-dimensional, incompressible flow field is given by $v = -Axy$, where v is in m/s, x and y are in meters, and A is a dimensional constant. There is no velocity component or variation in the z direction. Determine the dimensions of the constant, A. Find the simplest x component of velocity in this flow field. Calculate the acceleration of a fluid particle at point $(x, y) = (1, 2)$.

5.45 Consider the velocity field $\vec{V} = Ax/(x^2 + y^2)\hat{i} + Ay/(x^2 + y^2)\hat{j}$ in the xy plane, where $A = 10\ \text{m}^2/\text{s}$, and x and y are measured in meters. Is this an incompressible flow field? Derive an expression for the fluid acceleration. Evaluate the velocity and acceleration along the x axis, the y axis, and along a line defined by $y = x$. What can you conclude about this flow field?

5.46 An incompressible liquid with negligible viscosity flows steadily through a horizontal pipe of constant diameter. In a porous section of length $L = 0.3$ m, liquid is removed at a constant rate per unit length, so the uniform axial velocity in the pipe is $u(x) = U(1 - x/2L)$, where $U = 5$ m/s. Develop an expression for the acceleration of a fluid particle along the centerline of the porous section.

5.47 An incompressible liquid with negligible viscosity flows steadily through a horizontal pipe. The pipe diameter linearly varies from 4 in. to 1 in. over a length of 6 ft. Develop an expression for the acceleration of a fluid particle along the pipe centerline. Plot the centerline velocity and acceleration versus position along the pipe, if the inlet centerline velocity is 3 ft/s.

5.48 Consider the low-speed flow of air between parallel disks as shown. Assume that the flow is incompressible and inviscid, and that the velocity is purely radial and uniform at any section. The flow speed is $V = 15$ m/s at $R = 75$ mm. Simplify the continuity equation to a form applicable to this flow field. Show that a general expression for the velocity field is $\vec{V} = V(R/r)\hat{e}_r$ for $r_i \le r \le R$. Calculate the acceleration of a fluid particle at the locations $r = r_i$ and $r = R$.



P5.48

5.49 Solve Problem 4.123 to show that the radial velocity in the narrow gap is $V_r = Q/2\pi rh$. Derive an expression for the acceleration of a fluid particle in the gap.

5.50 As part of a pollution study, a model concentration c as a function of position x has been developed,

$$c(x) = A(e^{-x/2a} - e^{-x/a})$$

where $A = 3 \times 10^{-5}$ ppm (parts per million) and $a = 3$ ft. Plot this concentration from $x = 0$ to $x = 30$ ft. If a vehicle with a pollution sensor travels through the area at $u = U = 70$ ft/s, develop an expression for the measured concentration rate of change of c with time, and plot using the given data.

(a) At what location will the sensor indicate the most rapid rate of change?

(b) What is the value of this rate of change?

5.51 After a rainfall the sediment concentration at a certain point in a river increases at the rate of 100 parts per million (ppm) per hour. In addition, the sediment concentration increases with distance downstream as a result of influx from tributary streams; this rate of increase is 50 ppm per mile. At this point the stream flows at 0.5 mph. A boat is used to survey the sediment concentration. The operator is amazed to find three different apparent rates of change of sediment concentration when the boat travels upstream, drifts with the current, or travels downstream. Explain physically why the different rates are observed. If the speed of the boat is 2.5 mph, compute the three rates of change.

5.52 As an aircraft flies through a cold front, an onboard instrument indicates that ambient temperature drops at the rate of 0.7°F/min. Other instruments show an air speed of 400 knots and a 2500 ft/min rate of climb. The front is stationary and vertically uniform. Compute the rate of change of temperature with respect to horizontal distance through the cold front.

5.53 An aircraft flies due north at 300 mph ground speed. Its rate of climb is 3000 ft/min. The vertical temperature gradient is −3°F per 1000 ft of altitude. The ground temperature varies with position through a cold front, falling at the rate of 1°F per mile. Compute the rate of temperature change shown by a recorder on board the aircraft.

5.54 Wave flow of an incompressible fluid into a solid surface follows a sinusoidal pattern. Flow is axisymmetric about the z axis, which is normal to the surface. The z component of the flow follows the pattern

$$V_z = Az \sin\left(\frac{2\pi t}{T}\right)$$

Determine (a) the radial component of flow (V_r) and (b) the convective and local components of the acceleration vector.

5.55 Expand $(\vec{V}\cdot\nabla)\vec{V}$ in rectangular coordinates by direct substitution of the velocity vector to obtain the convective acceleration of a fluid particle. Verify the results given in Eqs. 5.11.

5.56 A steady, two-dimensional velocity field is given by $\vec{V} = Ax\hat{i} - Ay\hat{j}$, where $A = 1\ \mathrm{s}^{-1}$. Show that the streamlines for this flow are rectangular hyperbolas, $xy = C$. Obtain a general expression for the acceleration of a fluid particle in this velocity field. Calculate the acceleration of fluid particles at the points $(x, y) = (\frac{1}{2}, 2)$, $(1, 1)$, and $(2, \frac{1}{2})$, where x and y are measured in meters. Plot streamlines that correspond to $C = 0$, 1, and 2 m^2 and show the acceleration vectors on the streamline plot.

5.57 A velocity field is represented by the expression $\vec{V} = (Ax - B)\hat{i} - Ay\hat{j}$, where $A = 0.2\ \mathrm{s}^{-1}$, $B = 0.6\ \mathrm{m\cdot s^{-1}}$, and the coordinates are expressed in meters. Obtain a general expression for the acceleration of a fluid particle in this velocity field. Calculate the acceleration of fluid particles at points $(x, y) = (0, \frac{4}{3})$, $(1, 2)$, and $(2, 4)$. Plot a few streamlines in the xy plane. Show the acceleration vectors on the streamline plot.

5.58 A velocity field is represented by the expression $\vec{V} = (Ax - B)\hat{i} + Cy\hat{j} + Dt\hat{k}$, where $A = 2\ \mathrm{s}^{-1}$, $B = 4\ \mathrm{m\cdot s^{-1}}$, $D = 5\ \mathrm{m\cdot s^{-2}}$, and the coordinates are measured in meters. Determine the proper value for C if the flow field is to be incompressible. Calculate the acceleration of a fluid particle located at point $(x, y) = (3, 2)$. Plot a few flow streamlines in the xy plane.

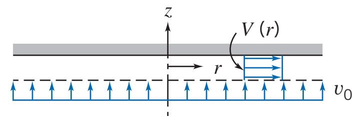
5.59 A linear approximate velocity profile was used in Problem 5.10 to model a laminar incompressible boundary layer on a flat plate. For this profile, obtain expressions for the x and y components of acceleration of a fluid particle in the boundary layer. Locate the maximum magnitudes of the x and y accelerations. Compute the ratio of the maximum x magnitude to the maximum y magnitude for the flow conditions of Problem 5.10.

5.60 A parabolic approximate velocity profile was used in Problem 5.11 to model flow in a laminar incompressible boundary layer on a flat plate. For this profile, find the x component of acceleration, a_x, of a fluid particle within the boundary layer. Plot a_x at location $x = 0.8$ m, where $\delta = 1.2$ mm, for a flow with $U = 6$ m/s. Find the maximum value of a_x at this x location.

5.61 Show that the velocity field of Problem 2.18 represents a possible incompressible flow field. Determine and plot the streamline passing through point $(x, y) = (2, 4)$ at $t = 1.5$ s. For the particle at the same point and time, show on the plot the velocity vector and the vectors representing the local, convective, and total accelerations.

5.62 A sinusoidal approximate velocity profile was used in Problem 5.12 to model flow in a laminar incompressible boundary layer on a flat plate. For this profile, obtain an expression for the x and y components of acceleration of a fluid particle in the boundary layer. Plot a_x and a_y at location $x = 3$ ft, where $\delta = 0.04$ in., for a flow with $U = 20$ ft/s. Find the maxima of a_x and a_y at this x location.

5.63 Air flows into the narrow gap, of height h, between closely spaced parallel disks through a porous surface as shown. Use a control volume, with outer surface located at position r, to show that the uniform velocity in the r direction is $V = v_0 r/2h$. Find an expression for the velocity component in the z direction ($v_0 \ll V$). Evaluate the components of acceleration for a fluid particle in the gap.



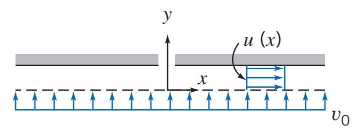
P5.63

5.64 The velocity field for steady inviscid flow from left to right over a circular cylinder, of radius R, is given by

$$\vec{V} = U\cos\theta\left[1 - \left(\frac{R}{r}\right)^2\right]\hat{e}_r - U\sin\theta\left[1 + \left(\frac{R}{r}\right)^2\right]\hat{e}_\theta$$

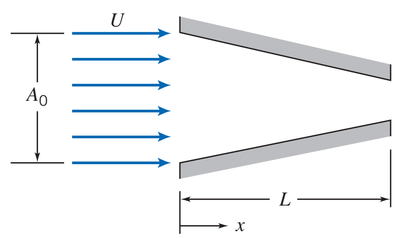
Obtain expressions for the acceleration of a fluid particle moving along the stagnation streamline ($\theta = \pi$) and for the acceleration along the cylinder surface ($r = R$). Plot a_r as a function of r/R for $\theta = \pi$, and as a function of θ for $r = R$; plot a_θ as a function of θ for $r = R$. Comment on the plots. Determine the locations at which these accelerations reach maximum and minimum values.

5.65 Air flows into the narrow gap, of height h, between closely spaced parallel plates through a porous surface as shown. Use a control volume, with outer surface located at position x, to show that the uniform velocity in the x direction is $u = v_0 x/h$. Find an expression for the velocity component in the y direction. Evaluate the acceleration of a fluid particle in the gap.



P5.65

5.66 Consider the incompressible flow of a fluid through a nozzle as shown. The area of the nozzle is given by $A = A_0(1 - bx)$ and the inlet velocity varies according to $U = U_0(0.5 + 0.5\cos\omega t)$ where $A_0 = 5\ \mathrm{ft}^2$, $L = 20$ ft, $b = 0.02\ \mathrm{ft}^{-1}$, $\omega = 0.16$ rad/s, and $U_0 = 20$ ft/s. Find and plot the acceleration on the centerline, with time as a parameter.

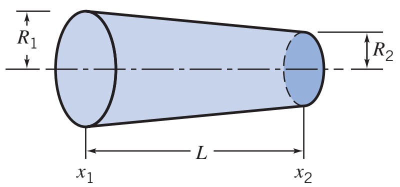


P5.66

5.67 Consider again the steady, two-dimensional velocity field of Problem 5.56. Obtain expressions for the particle coordinates, $x_p = f_1(t)$ and $y_p = f_2(t)$, as functions of time and

the initial particle position, (x_0, y_0) at $t = 0$. Determine the time required for a particle to travel from initial position, $(x_0, y_0) = (\frac{1}{2}, 2)$ to positions $(x, y) = (1, 1)$ and $(2, \frac{1}{2})$. Compare the particle accelerations determined by differentiating $f_1(t)$ and $f_2(t)$ with those obtained in Problem 5.56.

5.68 Consider the one-dimensional, incompressible flow through the circular channel shown. The velocity at section ① is given by $U = U_0 + U_1 \sin \omega t$, where $U_0 = 20$ m/s, $U_1 = 2$ m/s, and $\omega = 0.3$ rad/s. The channel dimensions are $L = 1$ m, $R_1 = 0.2$ m, and $R_2 = 0.1$ m. Determine the particle acceleration at the channel exit. Plot the results as a function of time over a complete cycle. On the same plot, show the acceleration at the channel exit if the channel is constant area, rather than convergent, and explain the difference between the curves.



P5.68

5.69 Which, if any, of the following flow fields are irrotational?
(a) $u = 2x^2 + y^2 - x^2 y$; $v = x^3 + x(y^2 - 2y)$
(b) $u = 2xy - x^2 + y$; $v = 2xy - y^2 + x^2$
(c) $u = xt + 2y$; $v = xt^2 - yt$
(d) $u = (x + 2y)xt$; $v = -(2x + y)yt$

5.70 Expand $(\vec{V} \cdot \nabla)\vec{V}$ in cylindrical coordinates by direct substitution of the velocity vector to obtain the convective acceleration of a fluid particle. (Recall the hint in footnote 1 on page 178.) Verify the results given in Eqs. 5.12.

5.71 Consider again the sinusoidal velocity profile used to model the x component of velocity for a boundary layer in Problem 5.12. Neglect the vertical component of velocity. Evaluate the circulation around the contour bounded by $x = 0.4$ m, $x = 0.6$ m, $y = 0$, and $y = 8$ mm. What would be the results of this evaluation if it were performed 0.2 m further downstream? Assume $U = 0.5$ m/s.

5.72 Consider the velocity field for flow in a rectangular "corner," $\vec{V} = Ax\hat{i} - Ay\hat{j}$, with $A = 0.3\ \text{s}^{-1}$, as in Example 5.8. Evaluate the circulation about the unit square of Example 5.8.

5.73 A flow is represented by the velocity field $\vec{V} = (x^7 - 21x^5y^2 + 35x^3y^4 - 7xy^6)\hat{i} + (7x^6y - 35x^4y^3 + 21x^2y^5 - y^7)\hat{j}$. Determine if the field is (a) a possible incompressible flow and (b) irrotational.

5.74 Consider the two-dimensional flow field in which $u = Ax^2$ and $v = Bxy$, where $A = 1/2\ \text{ft}^{-1}\cdot\text{s}^{-1}$, $B = -1\ \text{ft}^{-1}\cdot\text{s}^{-1}$, and the coordinates are measured in feet. Show that the velocity field represents a possible incompressible flow. Determine the rotation at point $(x, y) = (1, 1)$. Evaluate the circulation about the "curve" bounded by $y = 0$, $x = 1$, $y = 1$, and $x = 0$.

5.75 Consider the two-dimensional flow field in which $u = Axy$ and $v = By^2$, where $A = 1\text{m}^{-1}\cdot\text{s}^{-1}$, $B = -\frac{1}{2}\text{m}^{-1}\cdot\text{s}^{-1}$, and the coordinates are measured in meters. Show that the velocity field represents a possible incompressible flow. Determine the rotation at point $(x, y) = (1, 1)$. Evaluate the circulation about the "curve" bounded by $y = 0$, $x = 1$, $y = 1$, and $x = 0$.

***5.76** Consider a flow field represented by the stream function $\psi = 3x^5y - 10x^3y^3 + 3xy^5$. Is this a possible two-dimensional incompressible flow? Is the flow irrotational?

***5.77** Consider the flow field represented by the stream function $\psi = x^6 - 15x^4y^2 + 15x^2y^4 - y^6$. Is this a possible two-dimensional, incompressible flow? Is the flow irrotational?

***5.78** Consider a velocity field for motion parallel to the x axis with constant shear. The shear rate is $du/dy = A$, where $A = 0.1\ \text{s}^{-1}$. Obtain an expression for the velocity field, $\vec{V}$. Calculate the rate of rotation. Evaluate the stream function for this flow field.

***5.79** Consider a flow field represented by the stream function $\psi = -A/2(x^2 + y^2)$, where A = constant. Is this a possible two-dimensional incompressible flow? Is the flow irrotational?

***5.80** Consider the flow field represented by the stream function $\psi = Axy + Ay^2$, where $A = 1\ \text{s}^{-1}$. Show that this represents a possible incompressible flow field. Evaluate the rotation of the flow. Plot a few streamlines in the upper half plane.

***5.81** A flow field is represented by the stream function $\psi = x^2 - y^2$. Find the corresponding velocity field. Show that this flow field is irrotational. Plot several streamlines and illustrate the velocity field.

***5.82** Consider the velocity field given by $\vec{V} = Ax^2\hat{i} + Bxy\hat{j}$, where $A = 1\ \text{ft}^{-1}\cdot\text{s}^{-1}$, $B = -2\ \text{ft}^{-1}\cdot\text{s}^{-1}$, and the coordinates are measured in feet.
(a) Determine the fluid rotation.
(b) Evaluate the circulation about the "curve" bounded by $y = 0$, $x = 1$, $y = 1$, and $x = 0$.
(c) Obtain an expression for the stream function.
(d) Plot several streamlines in the first quadrant.

***5.83** Consider the flow represented by the velocity field $\vec{V} = (Ay + B)\hat{i} + Ax\hat{j}$, where $A = 10\ \text{s}^{-1}$, $B = 10$ ft/s, and the coordinates are measured in feet.
(a) Obtain an expression for the stream function.
(b) Plot several streamlines (including the stagnation streamline) in the first quadrant.
(c) Evaluate the circulation about the "curve" bounded by $y = 0$, $x = 1$, $y = 1$, and $x = 0$.

5.84 Consider again the viscometric flow of Example 5.7. Evaluate the average rate of rotation of a pair of perpendicular line segments oriented at ±45° from the x axis. Show that this is the same as in the example.

5.85 Consider the pressure-driven flow between stationary parallel plates separated by distance b. Coordinate y is measured from the bottom plate. The velocity field is given by $u = U(y/b)[1 - (y/b)]$. Obtain an expression for the circulation about a closed contour of height h and length L. Evaluate when $h = b/2$ and when $h = b$. Show that the same result is obtained from the area integral of the Stokes Theorem (Eq. 5.18).

***5.86** The velocity field near the core of a tornado can be approximated as

*These problems require material from sections that may be omitted without loss of continuity in the text material.

$$\vec{V} = -\frac{q}{2\pi r}\hat{e}_r + \frac{K}{2\pi r}\hat{e}_\theta$$

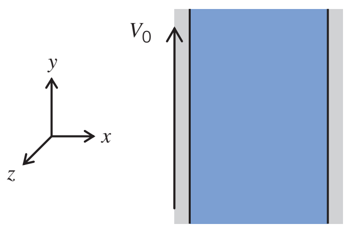
Is this an irrotational flow field? Obtain the stream function for this flow.

5.87 The velocity profile for fully developed flow in a circular tube is $V_z = V_{max}[1 - (r/R)^2]$. Evaluate the rates of linear and angular deformation for this flow. Obtain an expression for the vorticity vector, $\vec{\zeta}$.

5.88 Consider the pressure-driven flow between stationary parallel plates separated by distance $2b$. Coordinate y is measured from the channel centerline. The velocity field is given by $u = u_{max}[1 - (y/b)^2]$. Evaluate the rates of linear and angular deformation. Obtain an expression for the vorticity vector, $\vec{\zeta}$. Find the location where the vorticity is a maximum.

Momentum Equation

5.89 Consider a steady, laminar, fully developed, incompressible flow between two infinite plates, as shown. The flow is due to the motion of the left plate as well a pressure gradient that is applied in the y direction. Given the conditions that $\vec{V} \neq \vec{V}(z)$, $w = 0$, and that gravity points in the negative y direction, prove that $u = 0$ and that the pressure gradient in the y direction must be constant.



P5.89

5.90 Assume the liquid film in Example 5.9 is not isothermal, but instead has the following distribution:

$$T(y) = T_0 + (T_w - T_0)\left(1 - \frac{y}{h}\right)$$

where T_0 and T_w are, respectively, the ambient temperature and the wall temperature. The fluid viscosity decreases with increasing temperature and is assumed to be described by

$$\mu = \frac{\mu_0}{1 + a(T - T_0)}$$

with $a > 0$. In a manner similar to Example 5.9, derive an expression for the velocity profile.

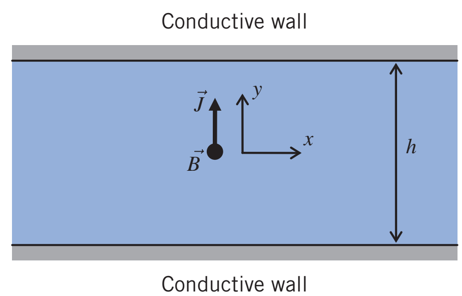
5.91 The x component of velocity in a laminar boundary layer in water is approximated as $u = U\sin(\pi y/2\delta)$, where $U = 3$ m/s and $\delta = 2$ mm. The y component of velocity is much smaller than u. Obtain an expression for the net shear force per unit volume in the x direction on a fluid element. Calculate its maximum value for this flow.

5.92 A linear velocity profile was used to model flow in a laminar incompressible boundary layer in Problem 5.10. Express the rotation of a fluid particle. Locate the maximum rate of rotation. Express the rate of angular deformation for a fluid particle. Locate the maximum rate of angular deformation. Express the rates of linear deformation for a fluid particle. Locate the maximum rates of linear deformation. Express the shear force per unit volume in the x direction. Locate the maximum shear force per unit volume; interpret this result.

5.93 Problem 4.35 gave the velocity profile for fully developed laminar flow in a circular tube as $u = u_{max}[1 - (r/R)^2]$. Obtain an expression for the shear force per unit volume in the x direction for this flow. Evaluate its maximum value for the conditions of Problem 4.35.

5.94 Assume the liquid film in Example 5.9 is horizontal (i.e., $\theta = 0°$) and that the flow is driven by a constant shear stress on the top surface ($y = h$), $\tau_{yx} = C$. Assume that the liquid film is thin enough and flat and that the flow is fully developed with zero net flow rate (flow rate $Q = 0$). Determine the velocity profile $u(y)$ and the pressure gradient dp/dx.

5.95 Consider a planar microchannel of width h, as shown (it is actually very long in the x direction and open at both ends). A Cartesian coordinate system with its origin positioned at the center of the microchannel is used in the study. The microchannel is filled with a weakly conductive solution. When an electric current is applied across the two conductive walls, the current density, $\vec{J}$, transmitted through the solution is parallel to the y axis. The entire device is placed in a constant magnetic field, $\vec{B}$, which is pointed outward from the plane (the z direction), as shown. Interaction between the current density and the magnetic field induces a *Lorentz force* of density $\vec{J} \times \vec{B}$. Assume that the conductive solution is incompressible, and since the sample volume is very small in lab-on-a-chip applications, the gravitational body force is neglected. Under steady state, the flow driven by the Lorentz force is described by the continuity (Eq. 5.1a) and Navier–Stokes equations (Eqs. 5.27), except the x, y, and z components of the latter have extra Lorentz force components on the right. Assuming that the flow is fully developed and the velocity field $\vec{V}$ is a function of y only, find the three components of velocity.



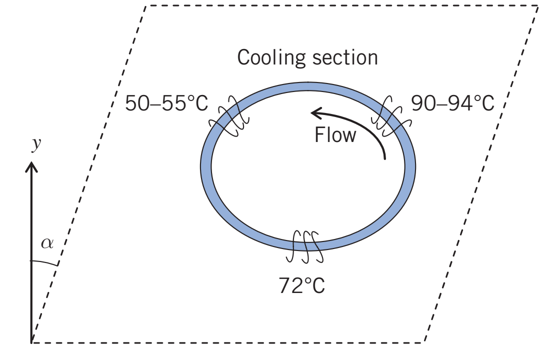
P5.95

5.96 The common thermal polymerase chain reaction (PCR) process requires the cycling of reagents through three distinct temperatures for denaturation (90–94°C), annealing (50–55°C), and extension (72°C). In continuous-flow PCR reactors, the temperatures of the three thermal zones are maintained as fixed while the reagents are cycled continuously through these zones. These temperature variations induce significant variations in the fluid density, which under appropriate conditions can be used to generate fluid motion. The figure depicts a thermosiphon-based PCR device (Chen et al., 2004, *Analytical Chemistry*, 76, 3707–3715).

The closed loop is filled with PCR reagents. The plan of the loop is inclined at an angle α with respect to the vertical. The loop is surrounded by three heaters and coolers that maintain different temperatures.

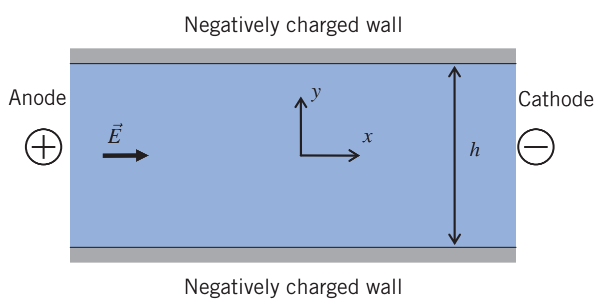
(a) Explain why the fluid automatically circulates in the closed loop along the counterclockwise direction.

(b) What is the effect of the angle α on the fluid velocity?



P5.96

5.97 Electro-osmotic flow (EOF) is the motion of liquid induced by an applied electric field across a charged capillary tube or microchannel. Assume the channel wall is negatively charged, a thin layer called the electric double layer (EDL) forms in the vicinity of the channel wall in which the number of positive ions is much larger than that of the negative ions. The net positively charged ions in the EDL then drag the electrolyte solution along with them and cause the fluid to flow toward the cathode. The thickness of the EDL is typically on the order of 10 nm. When the channel dimensions are much larger than the thickness of EDL, there is a slip velocity, $y - \frac{\varepsilon\zeta}{\mu}\vec{E}$, on the channel wall, where ε is the fluid permittivity, ζ is the negative surface electric potential, $\vec{E}$ is the electric field intensity, and μ is the fluid dynamic viscosity. Consider a microchannel formed by two parallel plates. The walls of the channel have a negative surface electric potential of ζ. The microchannel is filled with an electrolyte solution, and the microchannel ends are subjected to an electric potential difference that gives rise to a uniform electric field strength of E along the x direction. The pressure gradient in the channel is zero. Derive the velocity of the steady, fully developed electro-osmotic flow. Compare the velocity profile of the EOF to that of pressure-driven flow. Calculate the EOF velocity using $\varepsilon = 7.08 \times 10^{-10}$ C·V^{-1}m^{-1}, $\zeta = -0.1$ V, $\mu = 10^{-3}$ Pa·s, and $E = 1000$ V/m.



P5.97

Introduction to Computational Fluid Dynamics

***5.98** A tank contains water (20°C) at an initial depth $y_0 = 1$ m. The tank diameter is $D = 250$ mm, and a tube of diameter $d = 3$ mm and length $L = 4$ m is attached to the bottom of the tank. For laminar flow a reasonable model for the water level over time is

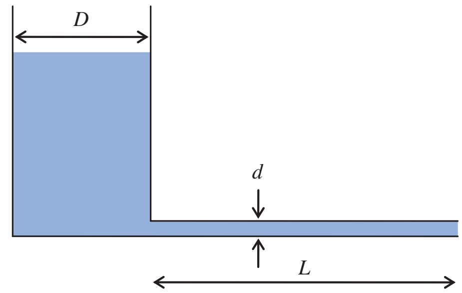
$$\frac{dy}{dt} = -\frac{d^4 \rho g}{32 D^2 \mu L} y \qquad y(0) = y_0$$

Using Euler methods with time steps of 12 min and 6 min:

(a) Estimate the water depth after 120 min, and compute the errors compared to the exact solution

$$y_{\text{exact}}(t) = y_0 e^{-\frac{d^4 \rho g}{32 D^2 \mu L} t}$$

(b) Plot the Euler and exact results.



P5.98

***5.99** Use the Euler method to solve and plot

$$\frac{dy}{dx} = \cos(x) \qquad y(0) = 0$$

from $x = 0$ to $x = \pi/2$, using step sizes of $\pi/48$, $\pi/96$, and $\pi/144$. Also plot the exact solution,

$$y(x) = \sin(x)$$

and compute the errors at $x = \pi/2$ for the three Euler method solutions.

***5.100** Use *Excel* to generate the solution of Eq. 5.31 for $m = 1$ shown in Fig. 5.18. To do so, you need to learn how to perform linear algebra in *Excel*. For example, for $N = 4$ you will end up with the matrix equation of Eq. 5.37. To solve this equation for the u values, you will have to compute the inverse of the 4×4 matrix, and then multiply this inverse into the 4×1 matrix on the right of the equation. In *Excel*, to do *array operations*, you must use the following rules: Pre-select the cells that will contain the result; use the appropriate *Excel array function* (look at *Excel's* Help for details); press Ctrl+Shift+Enter, not just Enter. For example, to invert the 4×4 matrix you would: Pre-select a blank 4×4 array that will contain the inverse matrix; type = *minverse*([array containing matrix to be inverted]); press Ctrl+Shift+Enter. To multiply a 4×4 matrix into a 4×1 matrix you would: Pre-select a blank 4×1 array that will contain the result; type = *mmult*([array containing 4×4 matrix], [array containing 4×1 matrix]); press Ctrl+Shift+Enter.

*These problems require material from sections that may be omitted without loss of continuity in the text material.

***5.101** Following the steps to convert the differential equation Eq. 5.31 (for $m = 1$) into a difference equation (for example, Eq. 5.37 for $N = 4$), solve

$$\frac{du}{dx} + u = 2\cos(2x) \qquad 0 \le x \le 1 \quad u(0) = 0$$

for $N = 4$, 8, and 16 and compare to the exact solution

$$u_{\text{exact}} = \frac{2}{5}\cos(2x) + \frac{4}{5}\sin(2x) - \frac{2}{5}e^{-x}$$

Hints: Follow the rules for *Excel* array operations as described in Problem 5.100. Only the right side of the difference equations will change, compared to the solution method of Eq. 5.31 (for example, only the right side of Eq. 5.37 needs modifying).

***5.102** Following the steps to convert the differential equation Eq. 5.31 (for $m = 1$) into a difference equation (for example, Eq. 5.37 for $N = 4$), solve

$$\frac{du}{dx} + u = 2x^2 + x \qquad 0 \le x \le 1 \quad u(0) = 3$$

for $N = 4$, 8, and 16 and compare to the exact solution

$$u_{\text{exact}} = 2x^2 - 3x + 3$$

Hint: Follow the hints provided in Problem 5.101.

***5.103** A 50-mm cube of mass $M = 3$ kg is sliding across an oiled surface. The oil viscosity is $\mu = 0.45$ N·s/m^2, and the thickness of the oil between the cube and surface is $\delta = 0.2$ mm. If the initial speed of the block is $u_0 = 1$ m/s, use the numerical method that was applied to the linear form of Eq. 5.31 to predict the cube motion for the first second of motion. Use $N = 4$, 8, and 16 and compare to the exact solution

$$u_{\text{exact}} = u_0 e^{-(A\mu/M\delta)t}$$

where A is the area of contact. *Hint:* Follow the hints provided in Problem 5.101.

***5.104** Use *Excel* to generate the solutions of Eq. 5.31 for $m = 2$, as shown in Fig. 5.21.

***5.105** Use *Excel* to generate the solutions of Eq. 5.31 for $m = 2$, as shown in Fig. 5.21, except use 16 points and as many iterations as necessary to obtain reasonable convergence.

***5.106** Use *Excel* to generate the solutions of Eq. 5.31 for $m = -1$, with $u(0) = 3$, using 4 and 16 points over the interval from $x = 0$ to $x = 3$, with sufficient iterations, and compare to the exact solution

$$u_{\text{exact}} = \sqrt{9 - 2x}$$

To do so, follow the steps described in "Dealing with Nonlinearity" section.

***5.107** An environmental engineer drops a pollution measuring probe with a mass of 0.3 slugs into a fast moving river (the speed of the water is $U = 25$ ft/s). The equation of motion for your speed u is

$$M\frac{du}{dt} = k(U - u)^2$$

where $k = 0.02$ lbf·s^2/ft^2 is a constant indicating the drag of the water. Use *Excel* to generate and plot the probe speed versus time (for the first 10 s) using the same approach as the solutions of Eq. 5.31 for $m = 2$, as shown in Fig 5.21, except use 16 points and as many iterations as necessary to obtain reasonable convergence. Compare your results to the exact solution

$$u_{\text{exact}} = \frac{kU^2 t}{M + kUt}$$

Hint: Use a substitution for $(U - u)$ so that the equation of motion looks similar to Eq. 5.31.

*These problems require material from sections that may be omitted without loss of continuity in the text material.

6

Incompressible Inviscid Flow

Case Study in Energy and the Environment

Wave Power: The *Limpet*

As we have discussed in previous *Case Studies in Energy and the Environment*, ocean waves contain a lot of energy; some regions of the world have an energy density (energy per width of water flow) of up to 75 kW/m in deep water, and up to 25 kW/m at the shoreline. Many ideas are being explored (some of which we have discussed earlier) for extracting this energy, from tethered buoys to articulated mechanisms. Technical issues are rapidly being resolved with many of these devices, but the Achilles heel of each of them is making the technologies work at a cost, for the power produced, that the consumer is willing to pay. Long-term, fossil fuels will become more expensive, and wave power will fall in cost, but we are not yet at the cross-over point. In the 1980s, wind power had the same kind of difficulty, but after several countries initially subsidized the industry, it is now becoming very

Two views of Wavegen's *Limpet* device (Pictures courtesy of Wavegen Ltd.)

cost-competitive. As with wind power, initial capital costs typically account for more than 90 percent of the cost of producing wave power; for fossil fuel plants the fuel supply itself is an ongoing part of the cost. To succeed, wave energy device developers have focused on driving down the initial capital costs.

The *Voith Hydro Wavegen Limited* company has been making big efforts in analyzing the costs and benefits of wave power with their *Limpet* (*L*and *I*nstalled *M*arine *P*owered *E*nergy *T*ransformer) device, shown in the photograph. This device was designed to be placed in onshore areas of high wave activity; in the long term, such devices will be designed for the higher-energy offshore regions. It is not a particularly impressive-looking device, but it has some interesting features. It looks like just a concrete block, but in fact is hollow and open to the sea on the underside, creating a trapped-air chamber; attached to it is an air turbine. It works pretty much like the swimming pool wave machine used at many amusement parks, except it runs in reverse. In these machines, air is blown in and out of a chamber beside the pool, which makes the water outside bob up and down, causing waves. For the Limpet, the arriving waves cause water in the chamber to rise and fall, which in turn compresses and expands the air trapped in the Limpet. If this is all we had, we would just have a device in which the water waves repeatedly compress and expand the trapped air. The clever innovation of the Limpet device is that a specially designed turbine is attached to the air chamber, so that the air flows through it first one way and then the other, extracting power. The *Wells* turbine (developed by Professor Alan Wells of Queen's University, Belfast) is a low-pressure air turbine that rotates continuously in one direction in spite of the direction of the air flow driving it. Its blades feature a symmetrical airfoil with its plane of symmetry in the plane of rotation and perpendicular to the air stream. Use of this bidirectional turbine allows power to be extracted as the air flows in *and* out of the chamber, avoiding the need for an expensive check valve system. The trade-off for the bidirectional turbine is that its efficiency is lower than that of a turbine with a constant air stream direction. However, the turbine is very simple and rugged: The blades are fixed onto the rotor and have no pitch-adjusting mechanism or gearbox and make no contact with the seawater. Turbines are discussed in some detail in Chapter 10, and some design concepts behind airfoil blade design in Chapter 9.

The whole device—concrete chamber, Wells turbine, and associated electronics—is rugged, inexpensive, and durable, so the goal of minimizing the initial capital cost is close to being realized. The technology used is called *OSW* (*o*scillating *w*ater *c*olumn). A new project involving the installation of 16 turbines into a breakwater off the coast of Spain is being constructed and is intended to supply green electricity to around 250 households with a rated power of nearly 300 kW.

In Chapter 5 we devoted a great deal of effort to deriving the differential equations (Eqs. 5.24) that describe the behavior of any fluid satisfying the continuum assumption. We also saw how these equations reduced to various particular forms—the most well known being the Navier–Stokes equations for an incompressible, constant viscosity fluid (Eqs. 5.27). Although Eqs. 5.27 describe the behavior of common fluids (e.g., water, air, lubricating oil) for a wide range of problems, as we discussed in

Chapter 5, they are unsolvable analytically except for the simplest of geometries and flows. For example, even using the equations to predict the motion of your coffee as you slowly stir it would require the use of an advanced computational fluid dynamics computer application, and the prediction would take a lot longer to compute than the actual stirring! In this chapter, instead of the Navier–Stokes equations, we will study Euler's equation, which applies to an inviscid fluid. Although truly inviscid fluids do not exist, many flow problems (especially in aerodynamics) can be successfully analyzed with the approximation that $\mu = 0$.

Momentum Equation for Frictionless Flow: Euler's Equation 6.1

Euler's equation (obtained from Eqs. 5.27 after neglecting the viscous terms) is

$$\rho \frac{D\vec{V}}{Dt} = \rho\vec{g} - \nabla p \tag{6.1}$$

This equation states that for an inviscid fluid the change in momentum of a fluid particle is caused by the body force (assumed to be gravity only) and the net pressure force. For convenience we recall that the particle acceleration is

$$\frac{D\vec{V}}{Dt} = \frac{\partial \vec{V}}{\partial t} + (\vec{V} \cdot \nabla)\vec{V} \tag{5.10}$$

In this chapter we will apply Eq. 6.1 to the solution of incompressible, inviscid flow problems. In addition to Eq. 6.1 we have the incompressible form of the mass conservation equation,

$$\nabla \cdot \vec{V} = 0 \tag{5.1c}$$

Equation 6.1 expressed in rectangular coordinates is

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = \rho g_x - \frac{\partial p}{\partial x} \tag{6.2a}$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = \rho g_y - \frac{\partial p}{\partial y} \tag{6.2b}$$

$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = \rho g_z - \frac{\partial p}{\partial z} \tag{6.2c}$$

If the z axis is assumed vertical, then $g_x = 0$, $g_y = 0$, and $g_z = -g$, so $\vec{g} = -g\hat{k}$.

In cylindrical coordinates, the equations in component form, with gravity the only body force, are

$$\rho a_r = \rho\left(\frac{\partial V_r}{\partial t} + V_r\frac{\partial V_r}{\partial r} + \frac{V_\theta}{r}\frac{\partial V_r}{\partial \theta} + V_z\frac{\partial V_r}{\partial z} - \frac{V_\theta^2}{r}\right) = \rho g_r - \frac{\partial p}{\partial r} \tag{6.3a}$$

$$\rho a_\theta = \rho\left(\frac{\partial V_\theta}{\partial t} + V_r\frac{\partial V_\theta}{\partial r} + \frac{V_\theta}{r}\frac{\partial V_\theta}{\partial \theta} + V_z\frac{\partial V_\theta}{\partial z} + \frac{V_r V_\theta}{r}\right) = \rho g_\theta - \frac{1}{r}\frac{\partial p}{\partial \theta} \tag{6.3b}$$

$$\rho a_z = \rho\left(\frac{\partial V_z}{\partial t} + V_r\frac{\partial V_z}{\partial r} + \frac{V_\theta}{r}\frac{\partial V_z}{\partial \theta} + V_z\frac{\partial V_z}{\partial z}\right) = \rho g_z - \frac{\partial p}{\partial z} \tag{6.3c}$$

If the z axis is directed vertically upward, then $g_r = g_\theta = 0$ and $g_z = -g$.

Equations 6.1, 6.2, and 6.3 apply to problems in which there are no viscous stresses. Before continuing with the main topic of this chapter (inviscid flow), let's consider for a moment when we have no viscous stresses, other than when $\mu = 0$. We recall from previous discussions that, in general, viscous stresses are present when we have fluid deformation (in fact this is how we initially defined a fluid); when we have no fluid deformation, i.e., when we have *rigid-body* motion, no viscous stresses will be present, even if $\mu \neq 0$. Hence Euler's equations apply to rigid-body motions as well as to inviscid flows. We discussed rigid-body motion in detail in Section 3.7 as a special case of fluid statics. As an exercise, you can show that Euler's equations can be used to solve Examples 3.9 and 3.10.

6.2 Euler's Equations in Streamline Coordinates

In Chapters 2 and 5 we pointed out that streamlines, drawn tangent to the velocity vectors at every point in the flow field, provide a convenient graphical representation. In steady flow a fluid particle will move along a streamline because, for steady flow, pathlines and streamlines coincide. Thus, in describing the motion of a fluid particle in a steady flow, in addition to using orthogonal coordinates x, y, z, the distance along a streamline is a logical coordinate to use in writing the equations of motion. "Streamline coordinates" also may be used to describe unsteady flow. Streamlines in unsteady flow give a graphical representation of the instantaneous velocity field.

For simplicity, consider the flow in the yz plane shown in Fig. 6.1. We wish to write the equations of motion in terms of the coordinate s, distance along a streamline, and the coordinate n, distance normal to the streamline. The pressure at the center of the fluid element is p. If we apply Newton's second law in the direction s of the streamline, to the fluid element of volume $ds\,dn\,dx$, then neglecting viscous forces we obtain

$$\left(p - \frac{\partial p}{\partial s}\frac{ds}{2}\right)dn\;dx - \left(p + \frac{\partial p}{\partial s}\frac{ds}{2}\right)dn\;dx - \rho g \sin\beta\, ds\;dn\;dx = \rho a_s\, ds\;dn\;dx$$

where β is the angle between the tangent to the streamline and the horizontal, and a_s is the acceleration of the fluid particle along the streamline. Simplifying the equation, we obtain

$$-\frac{\partial p}{\partial s} - \rho g \sin\beta = \rho a_s$$

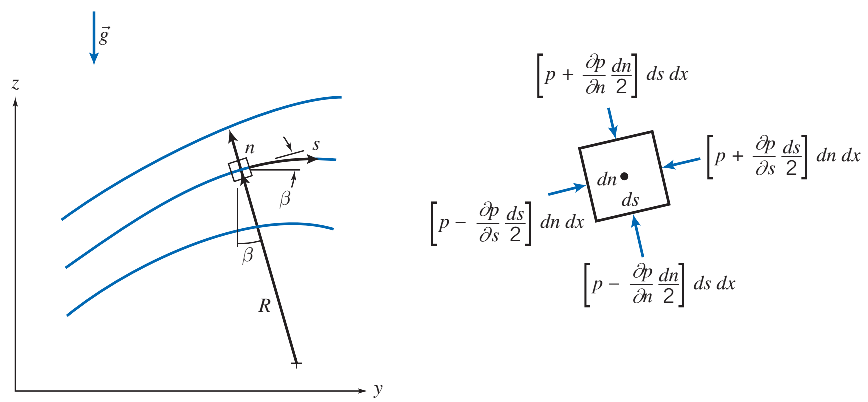


Fig. 6.1 Fluid particle moving along a streamline.

Since $\sin\beta = \partial z/\partial s$, we can write

$$-\frac{1}{\rho}\frac{\partial p}{\partial s} - g\frac{\partial z}{\partial s} = a_s$$

Along any streamline $V = V(s, t)$, and the material or total acceleration of a fluid particle in the streamwise direction is given by

$$a_s = \frac{DV}{Dt} = \frac{\partial V}{\partial t} + V\frac{\partial V}{\partial s}$$

Euler's equation in the streamwise direction with the z axis directed vertically upward is then

$$-\frac{1}{\rho}\frac{\partial p}{\partial s} - g\frac{\partial z}{\partial s} = \frac{\partial V}{\partial t} + V\frac{\partial V}{\partial s} \tag{6.4a}$$

For steady flow, and neglecting body forces, Euler's equation in the streamwise direction reduces to

$$\frac{1}{\rho}\frac{\partial p}{\partial s} = -V\frac{\partial V}{\partial s} \tag{6.4b}$$

which indicates that (for an incompressible, inviscid flow) *a decrease in velocity is accompanied by an increase in pressure* and conversely. This makes sense: The only force experienced by the particle is the net pressure force, so the particle accelerates toward low-pressure regions and decelerates when approaching high-pressure regions.

To obtain Euler's equation in a direction normal to the streamlines, we apply Newton's second law in the n direction to the fluid element. Again, neglecting viscous forces, we obtain

CLASSIC VIDEO

Pressure Fields and Fluid Acceleration.

$$\left(p - \frac{\partial p}{\partial n}\frac{dn}{2}\right)ds\,dx - \left(p + \frac{\partial p}{\partial n}\frac{dn}{2}\right)ds\,dx - \rho g\cos\beta\,dn\,dx\,ds = \rho a_n dn\,dx\,ds$$

where β is the angle between the n direction and the vertical, and a_n is the acceleration of the fluid particle in the n direction. Simplifying the equation, we obtain

$$-\frac{\partial p}{\partial n} - \rho g\cos\beta = \rho a_n$$

Since $\cos\beta = \partial z/\partial n$, we write

$$-\frac{1}{\rho}\frac{\partial p}{\partial n} - g\frac{\partial z}{\partial n} = a_n$$

The normal acceleration of the fluid element is toward the center of curvature of the streamline, in the minus n direction; thus in the coordinate system of Fig. 6.1, the familiar centripetal acceleration is written

$$a_n = -\frac{V^2}{R}$$

for steady flow, where R is the radius of curvature of the streamline at the point chosen. Then, Euler's equation normal to the streamline is written for steady flow as

$$\frac{1}{\rho}\frac{\partial p}{\partial n} + g\frac{\partial z}{\partial n} = \frac{V^2}{R} \tag{6.5a}$$

CLASSIC VIDEO

Pressure Fields and Fluid Acceleration.

For steady flow in a horizontal plane, Euler's equation normal to a streamline becomes

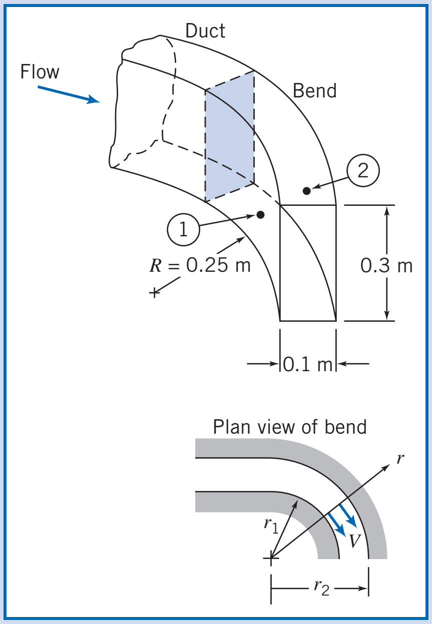
$$\frac{1}{\rho}\frac{\partial p}{\partial n} = \frac{V^2}{R} \tag{6.5b}$$

Equation 6.5b indicates that *pressure increases in the direction outward from the center of curvature of the streamlines*. This also makes sense: Because the only force experienced by the particle is the net pressure force, the pressure field creates the centripetal acceleration. In regions where the streamlines are straight, the radius of curvature, R, is infinite so *there is no pressure variation normal to straight streamlines*.

Example 6.1 FLOW IN A BEND

The flow rate of air at standard conditions in a flat duct is to be determined by installing pressure taps across a bend. The duct is 0.3 m deep and 0.1 m wide. The inner radius of the bend is 0.25 m. If the measured pressure difference between the taps is 40 mm of water, compute the approximate flow rate.

Given: Flow through duct bend as shown.



$$p_2 - p_1 = \rho_{H_2O} g\,\Delta h$$

where $\Delta h = 40$ mm H_2O. Air is at STP.

Find: Volume flow rate, Q.

Solution:
Apply Euler's n component equation across flow streamlines.

Governing equation: $\dfrac{\partial p}{\partial r} = \dfrac{\rho V^2}{r}$

Assumptions: (1) Frictionless flow.
(2) Incompressible flow.
(3) Uniform flow at measurement section.

For this flow, $p = p(r)$, so

$$\frac{\partial p}{\partial r} = \frac{dp}{dr} = \frac{\rho V^2}{r}$$

or

$$dp = \rho V^2 \frac{dr}{r}$$

Integrating gives

$$p_2 - p_1 = \rho V^2 \ln r \Big]_{r_1}^{r_2} = \rho V^2 \ln \frac{r_2}{r_1}$$

and hence

$$V = \left[\frac{p_2 - p_1}{\rho \ln(r_2/r_1)}\right]^{1/2}$$

But $\Delta p = p_2 - p_1 = \rho_{H_2O} g \Delta h$, so $V = \left[\dfrac{\rho_{H_2O} g \Delta h}{\rho \ln(r_2/r_1)}\right]^{1/2}$

Substituting numerical values,

$$V = \left[999 \frac{\text{kg}}{\text{m}^3} \times 9.81 \frac{\text{m}}{\text{s}^2} \times 0.04\ \text{m} \times \frac{\text{m}^3}{1.23\ \text{kg}} \times \frac{1}{\ln(0.35\ \text{m}/0.25\ \text{m})}\right]^{1/2}$$

$$= 30.8\ \text{m/s}$$

For uniform flow

$$Q = VA = 30.8 \frac{\text{m}}{\text{s}} \times 0.1\ \text{m} \times 0.3\ \text{m}$$

$$Q = 0.924\ \text{m}^3/\text{s} \longleftarrow Q$$

In this problem we assumed that the velocity is uniform across the section. In fact, the velocity in the bend approximates a free vortex (irrotational) profile in which $V \propto 1/r$ (where r is the radius) instead of $V = \text{const}$. Hence, this flow-measurement device could only be used to obtain approximate values of the flow rate (see Problem 6.32).

Bernoulli Equation: Integration of Euler's Equation Along a Streamline for Steady Flow 6.3

Compared to the viscous-flow equivalents, the momentum or Euler's equation for incompressible, inviscid flow (Eq. 6.1) is simpler mathematically, but solution (in conjunction with the mass conservation equation, Eq. 5.1c) still presents formidable difficulties in all but the most basic flow problems. One convenient approach for a steady flow is to integrate Euler's equation along a streamline. We will do this below using two different mathematical approaches, and each will result in the Bernoulli equation. Recall that in Section 4.4 we derived the Bernoulli equation by starting with a differential control volume; these two additional derivations will give us more insight into the restrictions inherent in use of the Bernoulli equation.

*Derivation Using Streamline Coordinates

Euler's equation for steady flow along a streamline (from Eq. 6.4a) is

$$-\frac{1}{\rho}\frac{\partial p}{\partial s} - g\frac{\partial z}{\partial s} = V\frac{\partial V}{\partial s} \tag{6.6}$$

If a fluid particle moves a distance, ds, along a streamline, then

$$\frac{\partial p}{\partial s} ds = dp \qquad \text{(the change in pressure along } s)$$

$$\frac{\partial z}{\partial s} ds = dz \qquad \text{(the change in elevation along } s)$$

$$\frac{\partial V}{\partial s} ds = dV \qquad \text{(the change in speed along } s)$$

Thus, after multiplying Eq. 6.6 by ds, we can write

$$-\frac{dp}{\rho} - g\,dz = V\,dV \qquad \text{or} \qquad \frac{dp}{\rho} + V\,dV + g\,dz = 0 \qquad (\text{along } s)$$

Integration of this equation gives

$$\int \frac{dp}{\rho} + \frac{V^2}{2} + gz = \text{constant} \quad (\text{along } s) \tag{6.7}$$

Before Eq. 6.7 can be applied, we must specify the relation between pressure and density. For the special case of incompressible flow, ρ = constant, and Eq. 6.7 becomes the Bernoulli equation,

$$\frac{p}{\rho} + \frac{V^2}{2} + gz = \text{constant} \tag{6.8}$$

Restrictions: (1) Steady flow.
(2) Incompressible flow.
(3) Frictionless flow.
(4) Flow along a streamline.

The Bernoulli equation is probably the most famous, and abused, equation in all of fluid mechanics. It is always tempting to use because it is a simple algebraic equation for relating the pressure, velocity, and elevation in a fluid. For example, it is used to explain the lift of a wing: In aerodynamics the gravity term is usually negligible, so Eq. 6.8 indicates that wherever the velocity is relatively high (e.g., on the upper surface of a wing), the pressure must be relatively low, and wherever the velocity is relatively low (e.g., on the lower surface of a wing), the pressure must be relatively high, generating substantial lift. Equation 6.8 indicates that, in general (if the flow is not constrained in some way), if a particle increases its elevation ($z \uparrow$) or moves into a higher pressure region ($p \uparrow$), it will tend to decelerate ($V \downarrow$); this makes sense from a momentum point of view (recall that the equation was derived from momentum considerations). These comments *only* apply if the four restrictions listed are reasonable. For example, Eq. 6.8 cannot be used to explain the pressure drop in a horizontal constant diameter pipe flow: according to it, for z = constant and V = constant, p = constant! We cannot stress enough that you should *keep the restrictions firmly in mind whenever you consider using the Bernoulli equation*! (In general, the Bernoulli constant in Eq. 6.8 has different values along different streamlines.[1])

*Derivation Using Rectangular Coordinates

The vector form of Euler's equation, Eq. 6.1, also can be integrated along a streamline. We shall restrict the derivation to steady flow; thus, the end result of our effort should be Eq. 6.7.

For steady flow, Euler's equation in rectangular coordinates can be expressed as

$$\frac{D\vec{V}}{Dt} = u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + w\frac{\partial \vec{V}}{\partial z} = (\vec{V} \cdot \nabla)\vec{V} = -\frac{1}{\rho}\nabla p - g\hat{k} \tag{6.9}$$

For steady flow the velocity field is given by $\vec{V} = \vec{V}(x, y, z)$. The streamlines are lines drawn in the flow field tangent to the velocity vector at every point. Recall again that for steady flow, streamlines, pathlines, and streaklines coincide. The motion of a

[1]For the case of irrotational flow, the constant has a single value throughout the entire flow field (Section 6.7).

*This section may be omitted without loss of continuity in the text material.

particle along a streamline is governed by Eq. 6.9. During time interval dt the particle has vector displacement $d\vec{s}$ along the streamline.

If we take the dot product of the terms in Eq. 6.9 with displacement $d\vec{s}$ along the streamline, we obtain a scalar equation relating pressure, speed, and elevation along the streamline. Taking the dot product of $d\vec{s}$ with Eq. 6.9 gives

$$(\vec{V}\cdot\nabla)\vec{V}\cdot d\vec{s} = -\frac{1}{\rho}\nabla p\cdot d\vec{s} - g\hat{k}\cdot d\vec{s} \tag{6.10}$$

where

$$d\vec{s} = dx\hat{i} + dy\hat{j} + dz\hat{k} \quad (\text{along } s)$$

Now we evaluate each of the three terms in Eq. 6.10, starting on the right,

$$\begin{aligned}
-\frac{1}{\rho}\nabla p\cdot d\vec{s} &= -\frac{1}{\rho}\left[\hat{i}\frac{\partial p}{\partial x} + \hat{j}\frac{\partial p}{\partial y} + \hat{k}\frac{\partial p}{\partial z}\right]\cdot[dx\hat{i} + dy\hat{j} + dz\hat{k}]\\
&= -\frac{1}{\rho}\left[\frac{\partial p}{\partial x}dx + \frac{\partial p}{\partial y}dy + \frac{\partial p}{\partial z}dz\right] \quad (\text{along } s)\\
-\frac{1}{\rho}\nabla p\cdot d\vec{s} &= -\frac{1}{\rho}dp \quad (\text{along } s)
\end{aligned}$$

and

$$\begin{aligned}
-g\hat{k}\cdot d\vec{s} &= -g\hat{k}\cdot[dx\hat{i} + dy\hat{j} + dz\hat{k}]\\
&= -g\,dz \qquad (\text{along } s)
\end{aligned}$$

Using a vector identity,[2] we can write the third term as

$$\begin{aligned}
(\vec{V}\cdot\nabla)\vec{V}\cdot d\vec{s} &= \left[\frac{1}{2}\nabla(\vec{V}\cdot\vec{V}) - \vec{V}\times(\nabla\times\vec{V})\right]\cdot d\vec{s}\\
&= \left\{\frac{1}{2}\nabla(\vec{V}\cdot\vec{V})\right\}\cdot d\vec{s} - \{\vec{V}\times(\nabla\times\vec{V})\}\cdot d\vec{s}
\end{aligned}$$

The last term on the right side of this equation is zero, since $\vec{V}$ is parallel to $d\vec{s}$ [recall from vector math that $\vec{V}\times(\nabla\times\vec{V})\cdot d\vec{s} = -(\nabla\times\vec{V})\times\vec{V}\cdot d\vec{s} = -(\nabla\times\vec{V})\cdot\vec{V}\times d\vec{s}$]. Consequently,

$$\begin{aligned}
(\vec{V}\cdot\nabla)\vec{V}\cdot d\vec{s} &= \frac{1}{2}\nabla(\vec{V}\cdot\vec{V})\cdot d\vec{s} = \frac{1}{2}\nabla(V^2)\cdot d\vec{s} \quad (\text{along } s)\\
&= \frac{1}{2}\left[\hat{i}\frac{\partial V^2}{\partial x} + \hat{j}\frac{\partial V^2}{\partial y} + \hat{k}\frac{\partial V^2}{\partial z}\right]\cdot[dx\hat{i} + dy\hat{j} + dz\hat{k}]\\
&= \frac{1}{2}\left[\frac{\partial V^2}{\partial x}dx + \frac{\partial V^2}{\partial y}dy + \frac{\partial V^2}{\partial z}dz\right]\\
(\vec{V}\cdot\nabla)\vec{V}\cdot d\vec{s} &= \frac{1}{2}d(V^2) \quad (\text{along } s)
\end{aligned}$$

[2]The vector identity

$$(\vec{V}\cdot\nabla)\vec{V} = \frac{1}{2}\nabla(\vec{V}\cdot\vec{V}) - \vec{V}\times(\nabla\times\vec{V})$$

may be verified by expanding each side into components.

Substituting these three terms into Eq. 6.10 yields

$$\frac{dp}{\rho} + \frac{1}{2}d(V^2) + g\,dz = 0 \quad \text{(along } s\text{)}$$

Integrating this equation, we obtain

$$\int \frac{dp}{\rho} + \frac{V^2}{2} + gz = \text{constant} \quad \text{(along } s\text{)}$$

If the density is constant, we obtain the Bernoulli equation

$$\frac{p}{\rho} + \frac{V^2}{2} + gz = \text{constant}$$

As expected, we see that the last two equations are identical to Eqs. 6.7 and 6.8 derived previously using streamline coordinates. The Bernoulli equation, derived using rectangular coordinates, is still subject to the restrictions: (1) steady flow, (2) incompressible flow, (3) frictionless flow, and (4) flow along a streamline.

Static, Stagnation, and Dynamic Pressures

The pressure, p, which we have used in deriving the Bernoulli equation, Eq. 6.8, is the thermodynamic pressure; it is commonly called the *static pressure*. The static pressure is the pressure experienced by the fluid particle as it moves (so it is something of a misnomer!)—we also have the stagnation and dynamic pressures, which we will define shortly. How do we measure the pressure in a fluid in motion?

In Section 6.2 we showed that there is no pressure variation normal to straight streamlines. This fact makes it possible to measure the static pressure in a flowing fluid using a wall pressure "tap," placed in a region where the flow streamlines are straight, as shown in Fig. 6.2*a*. The pressure tap is a small hole, drilled carefully in the wall, with its axis perpendicular to the surface. If the hole is perpendicular to the duct wall and free from burrs, accurate measurements of static pressure can be made by connecting the tap to a suitable pressure-measuring instrument [1].

In a fluid stream far from a wall, or where streamlines are curved, accurate static pressure measurements can be made by careful use of a static pressure probe, shown in Fig. 6.2*b*. Such probes must be designed so that the measuring holes are placed correctly with respect to the probe tip and stem to avoid erroneous results [2]. In use, the measuring section must be aligned with the local flow direction. (In these figures, it may appear that the pressure tap and small holes would allow flow to enter or leave or otherwise be entrained by the main flow, but each of these is ultimately attached to a pressure sensor or manometer and is therefore a dead-end, leading to no flow being possible—see Example 6.2.)

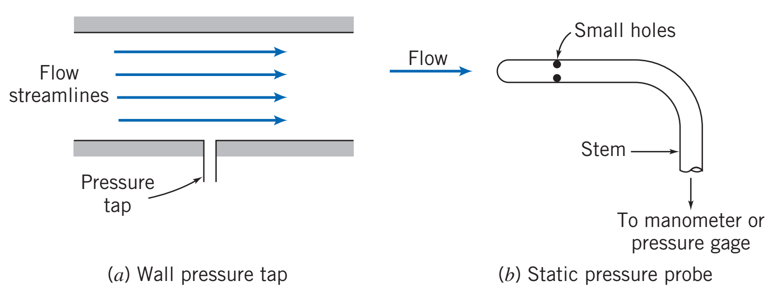


Fig. 6.2 Measurement of static pressure.

Static pressure probes, such as that shown in Fig 6.2*b*, and in a variety of other forms, are available commercially in sizes as small as 1.5 mm ($\frac{1}{16}$ in.) in diameter [3].

The *stagnation pressure* is obtained when a flowing fluid is decelerated to zero speed by a frictionless process. For incompressible flow, the Bernoulli equation can be used to relate changes in speed and pressure along a streamline for such a process. Neglecting elevation differences, Eq. 6.8 becomes

$$\frac{p}{\rho} + \frac{V^2}{2} = \text{constant}$$

If the static pressure is p at a point in the flow where the speed is V, then the stagnation pressure, p_0, where the stagnation speed, V_0, is zero, may be computed from

$$\frac{p_0}{\rho} + \cancelto{=0}{\frac{V_0^2}{2}} = \frac{p}{\rho} + \frac{V^2}{2}$$

or

$$p_0 = p + \frac{1}{2}\rho V^2 \tag{6.11}$$

Equation 6.11 is a mathematical statement of the definition of stagnation pressure, valid for incompressible flow. The term $\frac{1}{2}\rho V^2$ generally is called the *dynamic pressure*. Equation 6.11 states that the stagnation (or *total*) pressure equals the static pressure plus the dynamic pressure. One way to picture the three pressures is to imagine you are standing in a steady wind holding up your hand: The static pressure will be atmospheric pressure; the larger pressure you feel at the center of your hand will be the stagnation pressure; and the buildup of pressure (the difference between the stagnation and static pressures) will be the dynamic pressure. Solving Eq. 6.11 for the speed,

$$V = \sqrt{\frac{2(p_0 - p)}{\rho}} \tag{6.12}$$

Thus, if the stagnation pressure and the static pressure could be measured at a point, Eq. 6.12 would give the local flow speed.

Stagnation pressure is measured in the laboratory using a probe with a hole that faces directly upstream as shown in Fig. 6.3. Such a probe is called a stagnation pressure probe, or pitot (pronounced *pea-toe*) tube. Again, the measuring section must be aligned with the local flow direction.

We have seen that static pressure at a point can be measured with a static pressure tap or probe (Fig. 6.2). If we knew the stagnation pressure at the same point, then the flow speed could be computed from Eq. 6.12. Two possible experimental setups are shown in Fig. 6.4.

In Fig. 6.4*a*, the static pressure corresponding to point A is read from the wall static pressure tap. The stagnation pressure is measured directly at A by the total head tube,

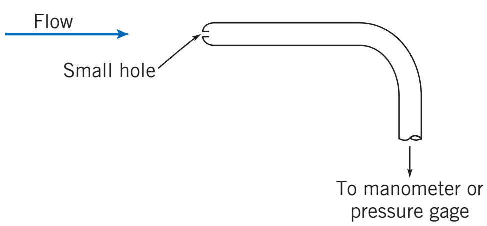


Fig. 6.3 Measurement of stagnation pressure.

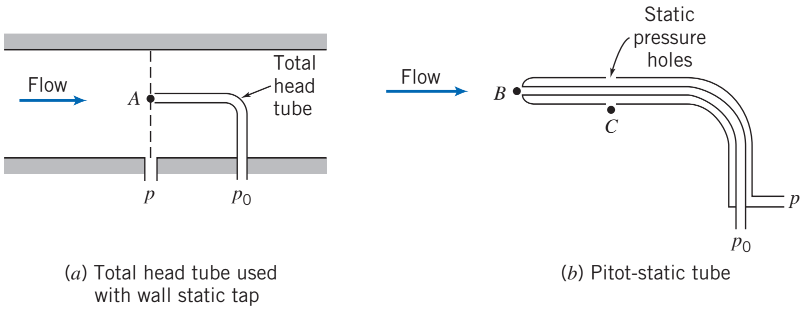


Fig. 6.4 Simultaneous measurement of stagnation and static pressures.

as shown. (The stem of the total head tube is placed downstream from the measurement location to minimize disturbance of the local flow.)

Two probes often are combined, as in the pitot-static tube shown in Fig. 6.4*b*. The inner tube is used to measure the stagnation pressure at point B, while the static pressure at C is sensed using the small holes in the outer tube. In flow fields where the static pressure variation in the streamwise direction is small, the pitot-static tube may be used to infer the speed at point B in the flow by assuming $p_B = p_C$ and using Eq. 6.12. (Note that when $p_B \neq p_C$, this procedure will give erroneous results.)

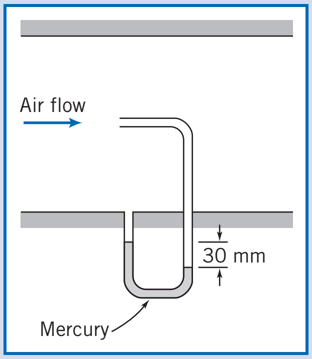
Remember that the Bernoulli equation applies only for incompressible flow (Mach number $M \leq 0.3$). The definition and calculation of the stagnation pressure for compressible flow will be discussed in Section 12.3.

Example 6.2 PITOT TUBE

A pitot tube is inserted in an air flow (at STP) to measure the flow speed. The tube is inserted so that it points upstream into the flow and the pressure sensed by the tube is the stagnation pressure. The static pressure is measured at the same location in the flow, using a wall pressure tap. If the pressure difference is 30 mm of mercury, determine the flow speed.

Given: A pitot tube inserted in a flow as shown. The flowing fluid is air and the manometer liquid is mercury.

Find: The flow speed.



Solution:

Governing equation: $\frac{p}{\rho} + \frac{V^2}{2} + gz = \text{constant}$

Assumptions: (1) Steady flow.
(2) Incompressible flow.
(3) Flow along a streamline.
(4) Frictionless deceleration along stagnation streamline.

Writing Bernoulli's equation along the stagnation streamline (with $\Delta z = 0$) yields

$$\frac{p_0}{\rho} = \frac{p}{\rho} + \frac{V^2}{2}$$

p_0 is the stagnation pressure at the tube opening where the speed has been reduced, without friction, to zero. Solving for V gives

$$V = \sqrt{\frac{2(p_0 - p)}{\rho_{\text{air}}}}$$

From the diagram,

$$p_0 - p = \rho_{\mathrm{Hg}} g h = \rho_{\mathrm{H_2O}} g h \mathrm{SG}_{\mathrm{Hg}}$$

and

$$V = \sqrt{\frac{2\rho_{\mathrm{H_2O}} g h \mathrm{SG}_{\mathrm{Hg}}}{\rho_{\mathrm{air}}}}$$

$$= \sqrt{2 \times 1000 \frac{\mathrm{kg}}{\mathrm{m}^3} \times 9.81 \frac{\mathrm{m}}{\mathrm{s}^2} \times 30\ \mathrm{mm} \times 13.6 \times \frac{\mathrm{m}^3}{1.23\ \mathrm{kg}} \times \frac{1\ \mathrm{m}}{1000\ \mathrm{mm}}}$$

$$V = 80.8\ \mathrm{m/s} \xleftarrow{\qquad\qquad\qquad\qquad\qquad\qquad V}$$

At $T = 20°\mathrm{C}$, the speed of sound in air is 343 m/s. Hence, $M = 0.236$ and the assumption of incompressible flow is valid.

This problem illustrates use of a pitot tube to determine flow speed. Pitot (or pitot-static) tubes are often placed on the exterior of aircraft to indicate air speed relative to the aircraft, and hence aircraft speed relative to the air.

Applications

The Bernoulli equation can be applied between any two points on a streamline provided that the other three restrictions are satisfied. The result is

$$\frac{p_1}{\rho} + \frac{V_1^2}{2} + g z_1 = \frac{p_2}{\rho} + \frac{V_2^2}{2} + g z_2 \tag{6.13}$$

where subscripts 1 and 2 represent any two points on a streamline. Applications of Eqs. 6.8 and 6.13 to typical flow problems are illustrated in Examples 6.3 through 6.5.

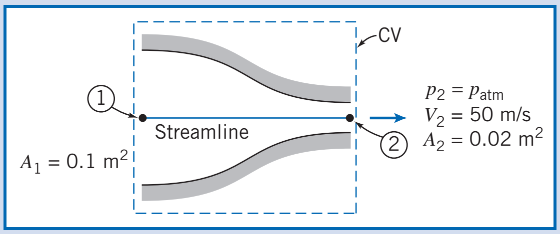
In some situations, the flow appears unsteady from one reference frame, but steady from another, which translates with the flow. Since the Bernoulli equation was derived by integrating Newton's second law for a fluid particle, it can be applied in any inertial reference frame (see the discussion of translating frames in Section 4.4). The procedure is illustrated in Example 6.6.

Example 6.3 NOZZLE FLOW

Air flows steadily at low speed through a horizontal *nozzle* (by definition a device for accelerating a flow), discharging to atmosphere. The area at the nozzle inlet is 0.1 m^2. At the nozzle exit, the area is 0.02 m^2. Determine the gage pressure required at the nozzle inlet to produce an outlet speed of 50 m/s.

Given: Flow through a nozzle, as shown.

Find: $p_1 - p_{\mathrm{atm}}$.



Solution:

Governing equations:

$$\frac{p_1}{\rho} + \frac{V_1^2}{2} + g z_1 = \frac{p_2}{\rho} + \frac{V_2^2}{2} + g z_2 \tag{6.13}$$

Continuity for incompressible and uniform flow:

$$\sum_{\mathrm{CS}} \vec{V} \cdot \vec{A} = 0 \tag{4.13b}$$

Assumptions: (1) Steady flow.
(2) Incompressible flow.
(3) Frictionless flow.
(4) Flow along a streamline.
(5) $z_1 = z_2$.
(6) Uniform flow at sections 1 and 2.

The maximum speed of 50 m/s is well below 100 m/s, which corresponds to Mach number $M \approx 0.3$ in standard air. Hence, the flow may be treated as incompressible.

Apply the Bernoulli equation along a streamline between points 1 and 2 to evaluate p_1. Then

$$p_1 - p_{\text{atm}} = p_1 - p_2 = \frac{\rho}{2}(V_2^2 - V_1^2)$$

Apply the continuity equation to determine V_1,

$$(-\rho V_1 A_1) + (\rho V_2 A_2) = 0 \qquad \text{or} \qquad V_1 A_1 = V_2 A_2$$

so that

$$V_1 = V_2 \frac{A_2}{A_1} = 50 \frac{\text{m}}{\text{s}} \times \frac{0.02\ \text{m}^2}{0.1\ \text{m}^2} = 10\ \text{m/s}$$

For air at standard conditions, $\rho = 1.23\ \text{kg/m}^3$. Then

$$p_1 - p_{\text{atm}} = \frac{\rho}{2}(V_2^2 - V_1^2)$$

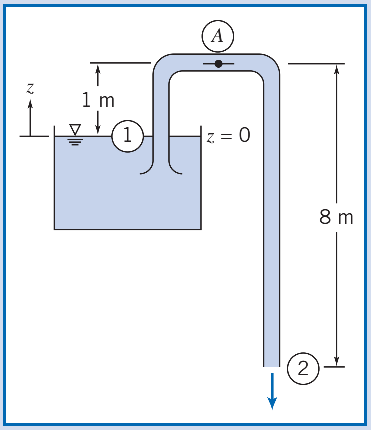
$$= \frac{1}{2} \times 1.23 \frac{\text{kg}}{\text{m}^3}\left[(50)^2 \frac{\text{m}^2}{\text{s}^2} - (10)^2 \frac{\text{m}^2}{\text{s}^2}\right] \frac{\text{N} \cdot \text{s}^2}{\text{kg} \cdot \text{m}}$$

$$p_1 - p_{\text{atm}} = 1.48\ \text{kPa} \longleftarrow p_1 - p_{\text{atm}}$$

Notes:
- ✓ This problem illustrates a typical application of the Bernoulli equation.
- ✓ The streamlines must be straight at the inlet and exit in order to have uniform pressures at those locations.

Example 6.4 FLOW THROUGH A SIPHON

A U-tube acts as a water siphon. The bend in the tube is 1 m above the water surface; the tube outlet is 7 m below the water surface. The water issues from the bottom of the siphon as a free jet at atmospheric pressure. Determine (after listing the necessary assumptions) the speed of the free jet and the minimum absolute pressure of the water in the bend.



Given: Water flowing through a siphon as shown.

Find: (a) Speed of water leaving as a free jet.
(b) Pressure at point A (the minimum pressure point) in the flow.

Solution:

Governing equation: $\dfrac{p}{\rho} + \dfrac{V^2}{2} + gz = \text{constant}$

Assumptions: (1) Neglect friction.
(2) Steady flow.
(3) Incompressible flow.
(4) Flow along a streamline.
(5) Reservoir is large compared with pipe.

Apply the Bernoulli equation between points 1 and 2.

$$\frac{p_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{p_2}{\rho} + \frac{V_2^2}{2} + gz_2$$

Since $\text{area}_{\text{reservoir}} \gg \text{area}_{\text{pipe}}$, then $V_1 \approx 0$. Also $p_1 = p_2 = p_{\text{atm}}$, so

$$gz_1 = \frac{V_2^2}{2} + gz_2 \qquad \text{and} \qquad V_2^2 = 2g(z_1 - z_2)$$

$$V_2 = \sqrt{2g(z_1 - z_2)} = \sqrt{2 \times 9.81 \frac{\text{m}}{\text{s}^2} \times 7\ \text{m}}$$

$$= 11.7\ \text{m/s} \longleftarrow V_2$$

To determine the pressure at location Ⓐ, we write the Bernoulli equation between ① and Ⓐ.

$$\frac{p_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{p_A}{\rho} + \frac{V_A^2}{2} + gz_A$$

Again $V_1 \approx 0$ and from conservation of mass $V_A = V_2$. Hence

$$\frac{p_A}{\rho} = \frac{p_1}{\rho} + gz_1 - \frac{V_2^2}{2} - gz_A = \frac{p_1}{\rho} + g(z_1 - z_A) - \frac{V_2^2}{2}$$

$$p_A = p_1 + \rho g(z_1 - z_A) - \rho\frac{V_2^2}{2}$$

$$= 1.01 \times 10^5 \frac{\text{N}}{\text{m}^2} + 999 \frac{\text{kg}}{\text{m}^3} \times 9.81 \frac{\text{m}}{\text{s}^2} \times (-1\ \text{m}) \frac{\text{N} \cdot \text{s}^2}{\text{kg} \cdot \text{m}}$$

$$-\frac{1}{2} \times 999 \frac{\text{kg}}{\text{m}^3} \times (11.7)^2 \frac{\text{m}^2}{\text{s}^2} \times \frac{\text{N} \cdot \text{s}^2}{\text{kg} \cdot \text{m}}$$

$$p_A = 22.8\ \text{kPa (abs) or } -78.5\ \text{kPa (gage)} \longleftarrow p_A$$

Notes:

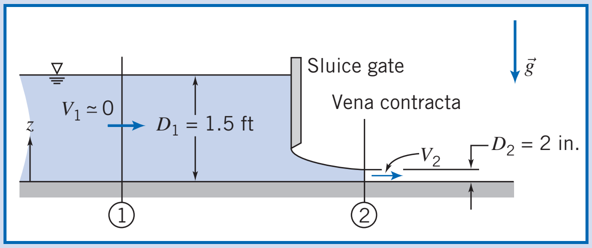
- ✓ This problem illustrates an application of the Bernoulli equation that includes elevation changes.
- ✓ It is interesting to note that when the Bernoulli equation applies between a reservoir and a free jet that it feeds at a location h below the reservoir surface, the jet speed will be $V = \sqrt{2gh}$; this is the same velocity a droplet (or stone) falling without friction from the reservoir level would attain if it fell a distance h. Can you explain why?
- ✓ Always take care when neglecting friction in any internal flow. In this problem, neglecting friction is reasonable if the pipe is smooth-surfaced and is relatively short. In Chapter 8 we will study frictional effects in internal flows.

Example 6.5 FLOW UNDER A SLUICE GATE

Water flows under a sluice gate on a horizontal bed at the inlet to a flume. Upstream from the gate, the water depth is 1.5 ft and the speed is negligible. At the vena contracta downstream from the gate, the flow streamlines are straight and the depth is 2 in. Determine the flow speed downstream from the gate and the discharge in cubic feet per second per foot of width.

Given: Flow of water under a sluice gate.

Find: (a) V_2.
(b) Q in ft^3/s/ft of width.



Solution:
Under the assumptions listed below, the flow satisfies all conditions necessary to apply the Bernoulli equation. The question is, what streamline do we use?

Governing equation: $\frac{p_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{p_2}{\rho} + \frac{V_2^2}{2} + gz_2$

Assumptions:
(1) Steady flow.
(2) Incompressible flow.
(3) Frictionless flow.
(4) Flow along a streamline.
(5) Uniform flow at each section.
(6) Hydrostatic pressure distribution (at each location, pressure increases linearly with depth).

If we consider the streamline that runs along the bottom of the channel ($z = 0$), because of assumption 6 the pressures at ① and ② are

$$p_1 = p_{atm} + \rho g D_1 \qquad \text{and} \qquad p_2 = p_{atm} + \rho g D_2$$

so that the Bernoulli equation for this streamline is

$$\frac{(p_{atm} + \rho g D_1)}{\rho} + \frac{V_1^2}{2} = \frac{(p_{atm} + \rho g D_2)}{\rho} + \frac{V_2^2}{2}$$

or

$$\frac{V_1^2}{2} + gD_1 = \frac{V_2^2}{2} + gD_2 \tag{1}$$

On the other hand, consider the streamline that runs along the free surface on both sides and down the inner surface of the gate. For this streamline

$$\frac{p_{atm}}{\rho} + \frac{V_1^2}{2} + gD_1 = \frac{p_{atm}}{\rho} + \frac{V_2^2}{2} + gD_2$$

or

$$\frac{V_1^2}{2} + gD_1 = \frac{V_2^2}{2} + gD_2 \tag{1}$$

We have arrived at the same equation (Eq. 1) for the streamline at the bottom and the streamline at the free surface, implying the Bernoulli constant is the same for both streamlines. We will see in Section 6.6 that this flow is one of a family of flows for which this is the case. Solving for V_2 yields

$$V_2 = \sqrt{2g(D_1 - D_2) + V_1^2}$$

But $V_1^2 \approx 0$, so

$$V_2 = \sqrt{2g(D_1 - D_2)} = \sqrt{2 \times 32.2 \frac{\text{ft}}{\text{s}^2} \times \left(1.5\ \text{ft} - 2\ \text{in.} \times \frac{\text{ft}}{12\ \text{in.}}\right)}$$

$$V_2 = 9.27\ \text{ft/s} \longleftarrow V_2$$

For uniform flow, $Q = VA = VDw$, or

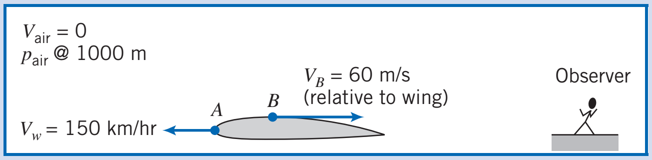
$$\frac{Q}{w} = VD = V_2 D_2 = 9.27 \frac{\text{ft}}{\text{s}} + 2\ \text{in.} \times \frac{\text{ft}}{12\ \text{in.}} = 1.55\ \text{ft}^2/\text{s}$$

$$\frac{Q}{w} = 1.55\ \text{ft}^3/\text{s/foot of width} \longleftarrow \frac{Q}{w}$$

Example 6.6 BERNOULLI EQUATION IN TRANSLATING REFERENCE FRAME

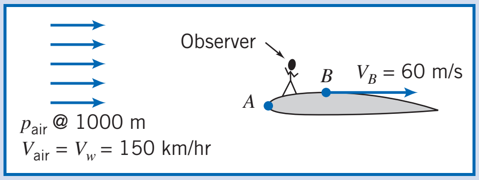
A light plane flies at 150 km/hr in standard air at an altitude of 1000 m. Determine the stagnation pressure at the leading edge of the wing. At a certain point close to the wing, the air speed *relative* to the wing is 60 m/s. Compute the pressure at this point.

Given: Aircraft in flight at 150 km/hr at 1000 m altitude in standard air.



Find: Stagnation pressure, p_{0_A}, at point A and static pressure, p_B, at point B.

Solution:
Flow is unsteady when observed from a fixed frame, that is, by an observer on the ground. However, an observer *on* the wing sees the following steady flow:



At $z = 1000$ m in standard air, the temperature is 281 K and the speed of sound is 336 m/s. Hence at point B, $M_B = V_B/c = 0.178$. This is less than 0.3, so the flow may be treated as incompressible. Thus the Bernoulli equation can be applied along a streamline in the moving observer's inertial reference frame.

Governing equation: $$\frac{p_{air}}{\rho} + \frac{V_{air}^2}{2} + gz_{air} = \frac{p_A}{\rho} + \frac{V_A^2}{2} + gz_A = \frac{p_B}{\rho} + \frac{V_B^2}{2} + gz_B$$

Assumptions:
(1) Steady flow.
(2) Incompressible flow ($V < 100$ m/s).
(3) Frictionless flow.
(4) Flow along a streamline.
(5) Neglect Δz.

Values for pressure and density may be found from Table A.3. Thus, at 1000 m, $p/p_{SL} = 0.8870$ and $\rho/\rho_{SL} = 0.9075$. Consequently,

$$p = 0.8870 p_{SL} = 0.8870 \times 1.01 \times 10^5 \frac{\text{N}}{\text{m}^2} = 8.96 \times 10^4 \text{ N/m}^2$$

and

$$\rho = 0.9075 \rho_{SL} = 0.9075 \times 1.23 \frac{\text{kg}}{\text{m}^3} = 1.12 \text{ kg/m}^3$$

Since the speed is $V_A = 0$ at the stagnation point,

$$p_{0_A} = p_{air} + \frac{1}{2}\rho V_{air}^2$$

$$= 8.96 \times 10^4 \frac{\text{N}}{\text{m}^2} + \frac{1}{2} \times 1.12 \frac{\text{kg}}{\text{m}^3}\left(150 \frac{\text{km}}{\text{hr}} \times 1000 \frac{\text{m}}{\text{km}} \times \frac{\text{hr}}{3600 \text{ s}}\right)^2 \times \frac{\text{N} \cdot \text{s}^2}{\text{kg} \cdot \text{m}}$$

$$p_{0_A} = 90.6 \text{ kPa(abs)} \longleftarrow p_{0_A}$$

Solving for the static pressure at B, we obtain

$$p_B = p_{\text{air}} + \frac{1}{2}\rho(V_{\text{air}}^2 - V_B^2)$$

$$p_B = 8.96 \times 10^4 \frac{\text{N}}{\text{m}^2} + \frac{1}{2} \times 1.12 \frac{\text{kg}}{\text{m}^3}\left[\left(150 \frac{\text{km}}{\text{hr}} \times 1000 \frac{\text{m}}{\text{km}} \times \frac{\text{hr}}{3600\ \text{s}}\right)^2 - (60)^2 \frac{\text{m}^2}{\text{s}^2}\right] \frac{\text{N} \cdot \text{s}^2}{\text{kg} \cdot \text{m}}$$

$$p_B = 88.6\ \text{kPa(abs)} \longleftarrow p_B$$

This problem gives a hint as to how a wing generates lift. The incoming air has a velocity $V_{\text{air}} = 150\ \text{km/hr} = 41.7\ \text{m/s}$ and *accelerates* to 60 m/s on the upper surface. This leads, through the Bernoulli equation, to a pressure *drop* of 1 kPa (from 89.6 kPa to 88.6 kPa). It turns out that the flow *decelerates* on the lower surface, leading to a pressure *rise* of about 1 kPa. Hence, the wing experiences a net upward pressure difference of about 2 kPa, a significant effect.

Cautions on Use of the Bernoulli Equation

In Examples 6.3 through 6.6, we have seen several situations where the Bernoulli equation may be applied because the restrictions on its use led to a reasonable flow model. However, in some situations you might be tempted to apply the Bernoulli equation where the restrictions are not satisfied. Some subtle cases that violate the restrictions are discussed briefly in this section.

Example 6.3 examined flow in a nozzle. In a *subsonic nozzle* (a converging section) the pressure drops, accelerating a flow. Because the pressure drops and the walls of the nozzle converge, there is no flow separation from the walls and the boundary layer remains thin. In addition, a nozzle is usually relatively short so frictional effects are not significant. All of this leads to the conclusion that the Bernoulli equation is suitable for use for subsonic nozzles.

CLASSIC VIDEO

Flow Visualization.

Sometimes we need to decelerate a flow. This can be accomplished using a *subsonic diffuser* (a diverging section), or by using a sudden expansion (e.g., from a pipe into a reservoir). In these devices the flow decelerates because of an adverse pressure gradient. As we discussed in Section 2.6, an adverse pressure gradient tends to lead to rapid growth of the boundary layer and its separation. Hence, we should be careful in applying the Bernoulli equation in such devices—at best, it will be an approximation. Because of area blockage caused by boundary-layer growth, pressure rise in actual diffusers always is less than that predicted for inviscid one-dimensional flow.

The Bernoulli equation was a reasonable model for the siphon of Example 6.4 because the entrance was well rounded, the bends were gentle, and the overall length was short. Flow separation, which can occur at inlets with sharp corners and in abrupt bends, causes the flow to depart from that predicted by a one-dimensional model and the Bernoulli equation. Frictional effects would not be negligible if the tube were long.

Example 6.5 presented an open-channel flow analogous to that in a nozzle, for which the Bernoulli equation is a good flow model. The hydraulic jump is an example of an open-channel flow with adverse pressure gradient. Flow through a hydraulic jump is mixed violently, making it impossible to identify streamlines. Thus the Bernoulli equation cannot be used to model flow through a hydraulic jump. We will see a more detailed presentation of open channel flows in Chapter 11.

The Bernoulli equation cannot be applied *through* a machine such as a propeller, pump, turbine, or windmill. The equation was derived by integrating along a stream

tube (Section 4.4) or a streamline (Section 6.3) in the absence of moving surfaces such as blades or vanes. It is impossible to have locally steady flow or to identify streamlines during flow through a machine. Hence, while the Bernoulli equation may be applied between points *before* a machine, or between points *after* a machine (assuming its restrictions are satisfied), it cannot be applied *through* the machine. (In effect, a machine will change the value of the Bernoulli constant.)

Finally, compressibility must be considered for flow of gases. Density changes caused by dynamic compression due to motion may be neglected for engineering purposes if the local Mach number remains below about $M \approx 0.3$, as noted in Examples 6.3 and 6.6. Temperature changes can cause significant changes in density of a gas, even for low-speed flow. Thus the Bernoulli equation could not be applied to air flow through a heating element (e.g., of a hand-held hair dryer) where temperature changes are significant.

CLASSIC VIDEO

Waves in Fluids and Stratified Flow.

6.4 The Bernoulli Equation Interpreted as an Energy Equation

The Bernoulli equation, Eq. 6.8, was obtained by integrating Euler's equation along a streamline for steady, incompressible, frictionless flow. Thus Eq. 6.8 was derived from the momentum equation for a fluid particle.

An equation identical in form to Eq. 6.8 (although requiring very different restrictions) may be obtained from the first law of thermodynamics. Our objective in this section is to reduce the energy equation to the form of the Bernoulli equation given by Eq. 6.8. Having arrived at this form, we then compare the restrictions on the two equations to help us understand more clearly the restrictions on the use of Eq. 6.8.

Consider steady flow in the absence of shear forces. We choose a control volume bounded by streamlines along its periphery. Such a boundary, shown in Fig. 6.5, often is called a *stream tube*.

Basic equation:

$$\dot{Q} - \overset{=0(1)}{\cancel{\dot{W}_s}} - \overset{=0(2)}{\cancel{\dot{W}_{\text{shear}}}} - \overset{=0(3)}{\cancel{\dot{W}_{\text{other}}}} = \overset{=0(4)}{\cancel{\frac{\partial}{\partial t}}}\int_{\text{CV}} e\,\rho\,d\mathcal{V} + \int_{\text{CS}} (e + pv)\,\rho\vec{V}\cdot d\vec{A} \qquad (4.56)$$

$$e = u + \frac{V^2}{2} + gz$$

Restrictions:
(1) $\dot{W}_s = 0$.
(2) $\dot{W}_{\text{shear}} = 0$.
(3) $\dot{W}_{\text{other}} = 0$.
(4) Steady flow.
(5) Uniform flow and properties at each section.

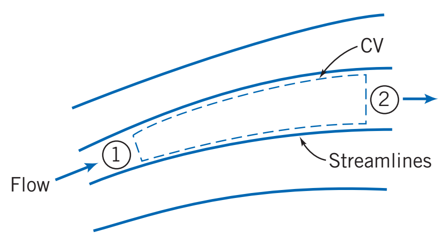


Fig. 6.5 Flow through a stream tube.

(Remember that here v represents the specific volume, and u represents the specific internal energy, not velocity!) Under these restrictions, Eq. 4.56 becomes

$$\left(u_1 + p_1 v_1 + \frac{V_1^2}{2} + gz_1\right)(-\rho_1 V_1 A_1) + \left(u_2 + p_2 v_2 + \frac{V_2^2}{2} + gz_2\right)(\rho_2 V_2 A_2) - \dot{Q} = 0$$

From continuity, with restrictions (4) and (5):

$$\sum_{\mathrm{CS}} \rho \vec{V} \cdot \vec{A} = 0 \tag{4.15b}$$

or

$$(-\rho_1 V_1 A_1) + (\rho_2 V_2 A_2) = 0$$

That is,

$$\dot{m} = \rho_1 V_1 A_1 = \rho_2 V_2 A_2$$

Also

$$\dot{Q} = \frac{\delta Q}{dt} = \frac{\delta Q}{dm}\frac{dm}{dt} = \frac{\delta Q}{dm}\dot{m}$$

Thus, from the energy equation, after rearranging

$$\left[\left(p_2 v_2 + \frac{V_2^2}{2} + gz_2\right) - \left(p_1 v_1 + \frac{V_1^2}{2} + gz_1\right)\right]\dot{m} + \left(u_2 - u_1 - \frac{\delta Q}{dm}\right)\dot{m} = 0$$

or

$$p_1 v_1 + \frac{V_1^2}{2} + gz_1 = p_2 v_2 + \frac{V_2^2}{2} + gz_2 + \left(u_2 - u_1 - \frac{\delta Q}{dm}\right)$$

Under the additional assumption (6) of incompressible flow, $v_1 = v_2 = 1/\rho$ and hence

$$\frac{p_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{p_2}{\rho} + \frac{V_2^2}{2} + gz_2 + \left(u_2 - u_1 - \frac{\delta Q}{dm}\right) \tag{6.14}$$

Equation 6.14 would reduce to the Bernoulli equation if the term in parentheses were zero. Thus, under the further restriction,

$$(7) \quad (u_2 - u_1 - \frac{\delta Q}{dm}) = 0$$

the energy equation reduces to

$$\frac{p_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{p_2}{\rho} + \frac{V_2^2}{2} + gz_2$$

or

$$\frac{p}{\rho} + \frac{V^2}{2} + gz = \text{constant} \tag{6.15}$$

Equation 6.15 is identical in form to the Bernoulli equation, Eq. 6.8. The Bernoulli equation was derived from momentum considerations (Newton's second law), and is valid for steady, incompressible, frictionless flow along a streamline. Equation 6.15 was obtained by applying the first law of thermodynamics to a stream tube control volume, subject to restrictions 1 through 7 above. Thus the Bernoulli equation (Eq. 6.8) and the identical form of the energy equation (Eq. 6.15) were developed from entirely different models, coming from entirely different basic concepts, and involving different restrictions.

It looks like we needed restriction (7) to finally transform the energy equation into the Bernoulli equation. In fact, we didn't! It turns out that for an incompressible and frictionless flow [restriction (6), and the fact we are looking only at flows with no shear forces], restriction (7) is automatically satisfied, as we will demonstrate in Example 6.7.

Example 6.7 INTERNAL ENERGY AND HEAT TRANSFER IN FRICTIONLESS INCOMPRESSIBLE FLOW

Consider frictionless, incompressible flow with heat transfer. Show that

$$u_2 - u_1 = \frac{\delta Q}{dm}$$

Given: Frictionless, incompressible flow with heat transfer.

Show: $u_2 - u_1 = \dfrac{\delta Q}{dm}$.

Solution:

In general, internal energy can be expressed as $u = u(T, v)$. For incompressible flow, $v =$ constant, and $u = u(T)$. Thus the thermodynamic state of the fluid is determined by the single thermodynamic property, T. For any process, the internal energy change, $u_2 - u_1$, depends only on the temperatures at the end states.

From the Gibbs equation, $Tds = du + \rho\ dv$, valid for a pure substance undergoing any process, we obtain

$$Tds = du$$

for incompressible flow, since $dv = 0$. Since the internal energy change, du, between specified end states, is independent of the process, we take a reversible process, for which $Tds = d(\delta Q/dm) = du$. Therefore,

$$u_2 - u_1 = \frac{\delta Q}{dm} \longleftarrow$$

For the steady, frictionless, and incompressible flow considered in this section, it is true that the first law of thermodynamics reduces to the Bernoulli equation. Each term in Eq. 6.15 has dimensions of energy per unit mass (we sometimes refer to the three terms in the equation as the "pressure" energy, kinetic energy, and potential energy per unit mass of the fluid). It is not surprising that Eq. 6.15 contains energy terms—after all, we used the first law of thermodynamics in deriving it. How did we end up with the same energy-like terms in the Bernoulli equation, which we derived from the momentum equation? The answer is because we integrated the momentum equation (which involves force terms) along a streamline (which involves distance), and by doing so ended up with work or energy terms (work being defined as force times distance): The work of gravity and pressure forces leads to a kinetic energy change (which came from integrating momentum over distance). In this context, we can think of the Bernoulli equation as a *mechanical energy balance*—the mechanical energy ("pressure" plus potential plus kinetic) will be constant. We must always bear in mind that for the Bernoulli equation to be valid along a streamline requires an incompressible inviscid flow, in addition to steady flow. It's interesting that these two properties of the flow—its compressibility and friction—are what "link" thermodynamic and mechanical energies. If a fluid is compressible, any flow-induced pressure changes will compress or expand the fluid, thereby doing work and changing the particle thermal energy; and friction, as we know from everyday experience, always converts mechanical to thermal energy. Their absence, therefore, breaks the link between the mechanical and thermal energies, and they are independent—it's as if they're in parallel universes!

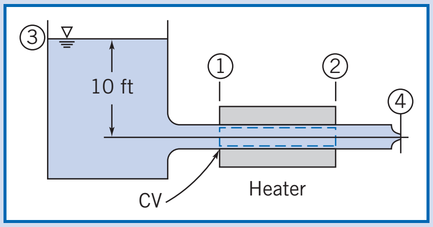
In summary, when the conditions are satisfied for the Bernoulli equation to be valid, we can consider separately the mechanical energy and the internal thermal energy of a fluid particle (this is illustrated in Example 6.8); when they are not satisfied, there will be an interaction between these energies, the Bernoulli equation becomes invalid, and we must use the full first law of thermodynamics.

Example 6.8 FRICTIONLESS FLOW WITH HEAT TRANSFER

Water flows steadily from a large open reservoir through a short length of pipe and a nozzle with cross-sectional area $A = 0.864$ in.^{2} A well-insulated 10 kW heater surrounds the pipe. Find the temperature rise of the water.

Given: Water flows from a large reservoir through the system shown and discharges to atmospheric pressure. The heater is 10 kW; $A_4 = 0.864$ in.^{2}

Find: The temperature rise of the water between points ① and ②.



Solution:

Governing equations:

$$\frac{p}{\rho} + \frac{V^2}{2} + gz = \text{constant} \tag{6.8}$$

$$\sum_{\text{CS}} \vec{V} \cdot \vec{A} = 0 \tag{4.13b}$$

$$\dot{Q} - \overset{=0(4)}{\cancel{\dot{W}_s}} - \overset{=0(4)}{\cancel{\dot{W}_{\text{shear}}}} = \overset{=0(1)}{\cancel{\frac{\partial}{\partial t}}} \int_{\text{CV}} e\,\rho\, d\forall + \int_{\text{CS}} \left(u + pv + \frac{V^2}{2} + gz\right) \rho \vec{V} \cdot d\vec{A} \tag{4.56}$$

Assumptions:
(1) Steady flow.
(2) Frictionless flow.
(3) Incompressible flow.
(4) No shaft work, no shear work.
(5) Flow along a streamline.
(6) Uniform flow at each section [a consequence of assumption (2)].

Under the assumptions listed, the first law of thermodynamics for the CV shown becomes

$$\dot{Q} = \int_{\text{CS}} \left(u + pv + \frac{V^2}{2} + gz\right) \rho \vec{V} \cdot d\vec{A}$$

$$= \int_{A_1} \left(u + pv + \frac{V^2}{2} + gz\right) \rho \vec{V} \cdot d\vec{A} + \int_{A_2} \left(u + pv + \frac{V^2}{2} + gz\right) \rho \vec{V} \cdot d\vec{A}$$

For uniform properties at ① and ②

$$\dot{Q} = -(\rho V_1 A_1)\left(u_1 + p_1 v + \frac{V_1^2}{2} + gz_1\right) + (\rho V_2 A_2)\left(u_2 + p_2 v + \frac{V_2^2}{2} + gz_2\right)$$

From conservation of mass, $\rho V_1 A_1 = \rho V_2 A_2 = \dot{m}$, so

$$\dot{Q} = \dot{m}\left[u_2 - u_1 + \left(\frac{p_2}{\rho} + \frac{V_2^2}{2} + gz_2\right) - \left(\frac{p_1}{\rho} + \frac{V_1^2}{2} + gz_1\right)\right]$$

For frictionless, incompressible, steady flow, along a streamline,

$$\frac{p}{\rho} + \frac{V^2}{2} + gz = \text{constant}$$

Therefore,

$$\dot{Q} = \dot{m}(u_2 - u_1)$$

Since, for an incompressible fluid, $u_2 - u_1 = c(T_2 - T_1)$, then

$$T_2 - T_1 = \frac{\dot{Q}}{\dot{m}c}$$

From continuity,

$$\dot{m} = \rho V_4 A_4$$

To find V_4, write the Bernoulli equation between the free surface at ③ and point ④.

$$\frac{p_3}{\rho} + \frac{V_3^2}{2} + gz_3 = \frac{p_4}{\rho} + \frac{V_4^2}{2} + gz_4$$

Since $p_3 = p_4$ and $V_3 \approx 0$, then

$$V_4 = \sqrt{2g(z_3 - z_4)} = \sqrt{2 \times 32.2 \frac{\text{ft}}{\text{s}^2} \times 10 \text{ ft}} = 25.4 \text{ ft/s}$$

and

$$\dot{m} = \rho V_4 A_4 = 1.94 \frac{\text{slug}}{\text{ft}^3} \times 25.4 \frac{\text{ft}}{\text{s}} \times 0.864 \text{ in.}^2 \times \frac{\text{ft}^2}{144 \text{ in.}^2} = 0.296 \text{ slug/s}$$

Assuming no heat loss to the surroundings, we obtain

$$T_2 - T_1 = \frac{\dot{Q}}{\dot{m}c} = 10 \text{ kW} \times 3413 \frac{\text{Btu}}{\text{kW} \cdot \text{hr}} \times \frac{\text{hr}}{3600 \text{ s}} \times \frac{\text{s}}{0.296 \text{ slug}} \times \frac{\text{slug}}{32.2 \text{ lbm}} \times \frac{\text{lbm} \cdot {}^\circ\text{R}}{1 \text{ Btu}}$$

$$T_2 - T_1 = 0.995\ {}^\circ\text{R} \xleftarrow{\qquad\qquad\qquad T_2 - T_1}$$

This problem illustrates that:

- ✓ In general, the first law of thermodynamics and the Bernoulli equation are independent equations.
- ✓ For an incompressible, inviscid flow the internal thermal energy is only changed by a heat transfer process, and is independent of the fluid mechanics.

Energy Grade Line and Hydraulic Grade Line 6.5

We have learned that for a steady, incompressible, frictionless flow, we may use the Bernoulli equation (Eq. 6.8), derived from the momentum equation, and also Eq. 6.15, derived from the energy equation:

$$\frac{p}{\rho} + \frac{V^2}{2} + gz = \text{constant} \tag{6.15}$$

We also interpreted the three terms comprised of "pressure," kinetic, and potential energies to make up the total mechanical energy, per unit mass, of the fluid. If we divide Eq. 6.15 by g, we obtain another form,

$$\frac{p}{\rho g} + \frac{V^2}{2g} + z = H \tag{6.16a}$$

Here H is the *total head* of the flow; it measures the total mechanical energy in units of meters or feet. We will learn in Chapter 8 that in a real fluid (one with friction) this head will *not* be constant but will continuously decrease in value as mechanical energy is converted to thermal; in this chapter H is constant. We can go one step further here and get a very useful graphical approach if we also define this to be the *energy grade line* (EGL),

$$EGL = \frac{p}{\rho g} + \frac{V^2}{2g} + z \tag{6.16b}$$

This can be measured using the pitot (total head) tube (shown in Fig. 6.3). Placing such a tube in a flow measures the total pressure, $p_0 = p + \frac{1}{2}\rho V^2$, so this will cause the height of a column *of the same fluid* to rise to a height $h = p_0/\rho g = p/\rho g + V^2/2g$. If the vertical location of the pitot tube is z, measured from some datum (e.g., the ground), the height of column of fluid *measured from the datum* will then be $h + z = p/\rho g + V^2/2g + z = EGL = H$. In summary, the height of the column, measured from the datum, attached to a pitot tube directly indicates the EGL.

We can also define the *hydraulic grade line* (HGL),

$$HGL = \frac{p}{\rho g} + z \tag{6.16c}$$

This can be measured using the static pressure tap (shown in Fig. 6.2*a*). Placing such a tube in a flow measures the static pressure, p, so this will cause the height of a column *of the same fluid* to rise to a height $h = p/\rho g$. If the vertical location of the tap is also at z, measured from some datum, the height of column of fluid *measured from the datum* will then be $h + z = p/\rho g + z = HGL$. The height of the column attached to a static pressure tap thus directly indicates the HGL.

From Eqs. 6.16b and 6.16c we obtain

$$EGL - HGL = \frac{V^2}{2g} \tag{6.16d}$$

which shows that the difference between the EGL and HGL is always the dynamic pressure term.

To see a graphical interpretation of the EGL and HGL, refer to the example shown in Fig. 6.6, which shows frictionless flow from a reservoir, through a pipe reducer.

At all locations the EGL is the same because there is no loss of mechanical energy. Station 1 is at the reservoir, and here the EGL and HGL coincide with the free surface: in Eqs. 6.16b and 6.16c $p = 0$ (gage), $V = 0$, and $z = z_1$, so $EGL_1 = HGL_1 = H = z_1$; all of the mechanical energy is potential. (If we were to place a pitot tube in the fluid at station 1, the fluid would of course just rise to the free surface level.)

At station 2 we have a pitot (total head) tube and a static head tap. The pitot tube's column indicates the correct value of the EGL ($EGL_1 = EGL_2 = H$), but *something* changed between the two stations: The fluid now has significant kinetic energy and has lost some potential energy (can you determine from the figure what happened to the pressure?). From Eq. 6.16d, we can see that the HGL is lower than the EGL by $V_2^2/2g$; the HGL at station 2 shows this.

From station 2 to station 3 there is a reduction in diameter, so continuity requires that $V_3 > V_2$; hence the gap between the EGL and HGL increases further, as shown.

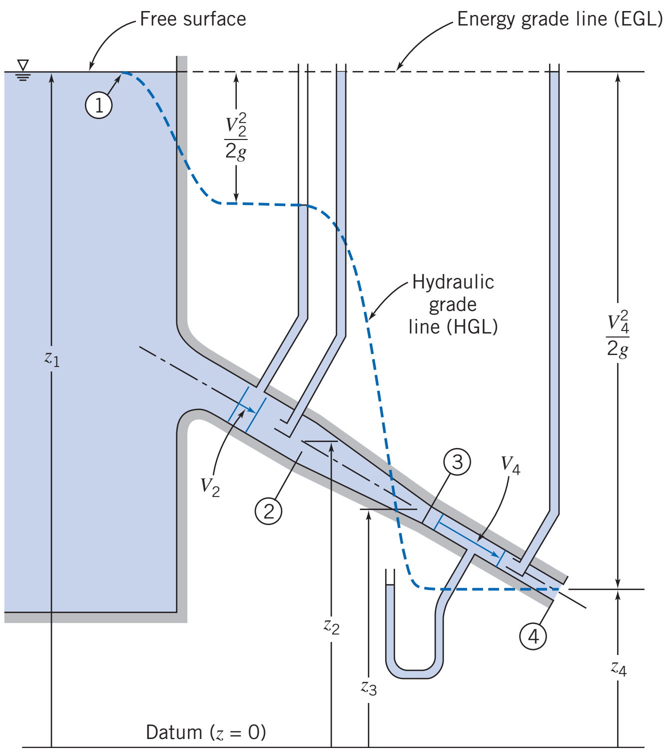


Fig. 6.6 Energy and hydraulic grade lines for frictionless flow.

Station 4 is at the exit (to the atmosphere). Here the pressure is zero (gage), so the EGL consists entirely of kinetic and potential energy terms, and $HGL_4 = HGL_3$. We can summarize two important ideas when sketching EGL and HGL curves:

1. The EGL is constant for incompressible, inviscid flow (in the absence of work devices). We will see in Chapter 8 that work devices may increase or decrease the EGL, and friction will always lead to a fall in the EGL.
2. The HGL is always lower than the EGL by distance $V^2/2g$. Note that the value of velocity V depends on the overall system (e.g., reservoir height, pipe diameter, etc.), but *changes* in velocity *only* occur when the diameter changes.

Unsteady Bernoulli Equation: Integration of Euler's Equation Along a Streamline (on the Web) 6.6*

Irrotational Flow 6.7*

We have already discussed irrotational flows in Section 5.3. These are flows in which the fluid particles do not rotate ($\vec{\omega} = 0$). We recall that the only stresses that can generate particle rotation are shear stresses; hence, inviscid flows (i.e., those with zero

*These sections may be omitted without loss of continuity in the text material. (Note that Section 5.2 contains background material needed for study of Section 6.7.)

shear stresses) will be irrotational, unless the particles were initially rotating. Using Eq. 5.14, we obtain the irrotationality condition

$$\nabla \times \vec{V} = 0 \tag{6.22}$$

leading to

$$\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z} = \frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 \tag{6.23}$$

In cylindrical coordinates, from Eq. 5.16, the irrotationality condition requires that

$$\frac{1}{r}\frac{\partial V_z}{\partial \theta} - \frac{\partial V_\theta}{\partial z} = \frac{\partial V_r}{\partial z} - \frac{\partial V_z}{\partial r} = \frac{1}{r}\frac{\partial r V_\theta}{\partial r} - \frac{1}{r}\frac{\partial V_r}{\partial \theta} = 0 \tag{6.24}$$

Bernoulli Equation Applied to Irrotational Flow

In Section 6.3, we integrated Euler's equation along a streamline for steady, incompressible, inviscid flow to obtain the Bernoulli equation

$$\frac{p}{\rho} + \frac{V^2}{2} + gz = \text{constant} \tag{6.8}$$

Equation 6.8 can be applied between any two points on the *same* streamline. In general, the value of the constant will vary from streamline to streamline.

If, in addition to being inviscid, steady, and incompressible, the flow field is also irrotational (i.e., the particles had no initial rotation), so that $\nabla \times \vec{V} = 0$ (Eq. 6.22), we can show that Bernoulli's equation can be applied between any and all points in the flow. Then the value of the constant in Eq. 6.8 is the same for all streamlines. To illustrate this, we start with Euler's equation in vector form,

$$(\vec{V} \cdot \nabla)\vec{V} = -\frac{1}{\rho}\nabla p - g\hat{k} \tag{6.9}$$

Using the vector identity

$$(\vec{V} \cdot \nabla)\vec{V} = \frac{1}{2}\nabla(\vec{V} \cdot \vec{V}) - \vec{V} \times (\nabla \times \vec{V})$$

we see for irrotational flow, where $\nabla \times \vec{V} = 0$, that

$$(\vec{V} \cdot \nabla)\vec{V} = \frac{1}{2}\nabla(\vec{V} \cdot \vec{V})$$

and Euler's equation for irrotational flow can be written as

$$\frac{1}{2}\nabla(\vec{V} \cdot \vec{V}) = \frac{1}{2}\nabla(V^2) = -\frac{1}{\rho}\nabla p - g\hat{k} \tag{6.25}$$

Consider a displacement in the flow field from position $\vec{r}$ to position $\vec{r} + d\vec{r}$; the displacement $d\vec{r}$ is an *arbitrary* infinitesimal displacement in *any* direction, not necessarily along a streamline. Taking the dot product of $d\vec{r} = dx\hat{i} + dy\hat{j} + dz\hat{k}$ with each of the terms in Eq. 6.25, we have

$$\frac{1}{2}\nabla(V^2) \cdot d\vec{r} = -\frac{1}{\rho}\nabla p \cdot d\vec{r} - g\hat{k} \cdot d\vec{r}$$

and hence

$$\frac{1}{2}d(V^2) = -\frac{dp}{\rho} - g\,dz$$

or

$$\frac{dp}{\rho} + \frac{1}{2}\,d(V^2) + g\,dz = 0$$

Integrating this equation for incompressible flow gives

$$\frac{p}{\rho} + \frac{V^2}{2} + gz = \text{constant} \tag{6.26}$$

Since $d\vec{r}$ was an arbitrary displacement, Eq. 6.26 is valid between *any* two points (i.e., not just along a streamline) in a steady, incompressible, inviscid flow that is also irrotational (see Example 6.5).

Velocity Potential

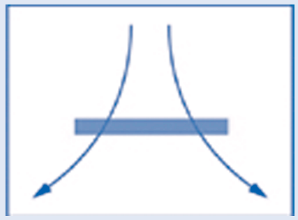
We were introduced in Section 5.2 to the notion of the stream function ψ for a two-dimensional incompressible flow.

For irrotational flow we can introduce a companion function, the *potential function* ϕ, defined by

$$\vec{V} = -\nabla\phi \tag{6.27}$$

VIDEO

An Example of Irrotational Flow.



Why this definition? Because it guarantees that *any* continuous scalar function $\phi(x, y, z, t)$ *automatically* satisfies the irrotationality condition (Eq. 6.22) because of a fundamental identity:[3]

$$\nabla \times \vec{V} = -\nabla \times \nabla\phi = -\text{curl}(\text{grad}\,\phi) \equiv 0 \tag{6.28}$$

The minus sign (used in most textbooks) is inserted simply so that ϕ decreases in the flow direction (analogous to the temperature decreasing in the direction of heat flow in heat conduction). Thus,

$$u = -\frac{\partial\phi}{\partial x}, \quad v = -\frac{\partial\phi}{\partial y}, \quad \text{and} \quad w = -\frac{\partial\phi}{\partial z} \tag{6.29}$$

(You can check that the irrotationality condition, Eq. 6.22, is satisfied identically.)

In cylindrical coordinates,

$$\nabla = \hat{e}_r\frac{\partial}{\partial r} + \hat{e}_\theta\frac{1}{r}\frac{\partial}{\partial\theta} + \hat{k}\frac{\partial}{\partial z} \tag{3.19}$$

From Eq. 6.27, then, in cylindrical coordinates

$$V_r = -\frac{\partial\phi}{\partial r} \qquad V_\theta = -\frac{1}{r}\frac{\partial\phi}{\partial\theta} \qquad V_z = -\frac{\partial\phi}{\partial z} \tag{6.30}$$

CLASSIC VIDEO

Vorticity.

Because $\nabla \times \nabla\phi \equiv 0$ for all ϕ, the velocity potential exists only for irrotational flow.

Irrotationality may be a valid assumption for those regions of a flow in which viscous forces are negligible. (For example, such a region exists outside the boundary layer in the flow over a wing surface, and can be analyzed to find the lift produced by the wing.) The theory for irrotational flow is developed in terms of an imaginary ideal fluid whose viscosity is identically zero. Since, in an irrotational flow, the velocity field may be defined by the potential function ϕ, the theory is often referred to as potential flow theory.

[3] That $\nabla \times \nabla(\ ) \equiv 0$ can easily be demonstrated by expanding into components.

All real fluids possess viscosity, but there are many situations in which the assumption of inviscid flow considerably simplifies the analysis and, at the same time, gives meaningful results. Because of its relative simplicity and mathematical beauty, potential flow has been studied extensively.[4]

Stream Function and Velocity Potential for Two-Dimensional, Irrotational, Incompressible Flow: Laplace's Equation

For a two-dimensional, incompressible, irrotational flow we have expressions for the velocity components, u and v, in terms of both the stream function ψ, and the velocity potential ϕ,

$$u = \frac{\partial \psi}{\partial y} \qquad v = -\frac{\partial \psi}{\partial x} \tag{5.4}$$

$$u = -\frac{\partial \phi}{\partial x} \qquad v = -\frac{\partial \phi}{\partial y} \tag{6.29}$$

Substituting for u and v from Eq. 5.4 into the irrotationality condition,

$$\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 \tag{6.23}$$

we obtain

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = \nabla^2 \psi = 0 \tag{6.31}$$

Substituting for u and v from Eq. 6.29 into the continuity equation,

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{5.3}$$

we obtain

$$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} = \nabla^2 \phi = 0 \tag{6.32}$$

Equations 6.31 and 6.32 are forms of Laplace's equation—an equation that arises in many areas of the physical sciences and engineering. Any function ψ or ϕ that satisfies Laplace's equation represents a possible two-dimensional, incompressible, irrotational flow field.

Table 6.1 summarizes the results of our discussion of the stream function and velocity potential for two dimensional flows.

The same rules (of when incompressibility and irrotationality apply, and with the appropriate form of Laplace's equation) are valid for the stream function and velocity potential when expressed in cylindrical coordinates,

$$V_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} \qquad \text{and} \qquad V_\theta = -\frac{\partial \psi}{\partial r} \tag{5.8}$$

[4]Anyone interested in a detailed study of potential flow theory may find [4–6] of interest.

Table 6.1

Definitions of ψ and ϕ, and Conditions Necessary for Satisfying Laplace's Equation

Definition	Always satisfies . . .	Satisfies Laplace equation . . . $\frac{\partial^2()}{\partial x^2} + \frac{\partial^2()}{\partial y^2} = \nabla^2() = 0$
Stream function ψ $u = \frac{\partial \psi}{\partial y} \quad v = -\frac{\partial \psi}{\partial x}$	. . . incompressibility: $\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = \frac{\partial^2 \psi}{\partial x \partial y} - \frac{\partial^2 \psi}{\partial y \partial x} \equiv 0$	. . . only if irrotational: $\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = -\frac{\partial^2 \psi}{\partial x \partial x} - \frac{\partial^2 \psi}{\partial y \partial y} = 0$
Velocity potential ϕ $u = -\frac{\partial \phi}{\partial x} \quad v = -\frac{\partial \phi}{\partial y}$	. . . irrotationality: $\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = -\frac{\partial^2 \phi}{\partial x \partial y} - \frac{\partial^2 \phi}{\partial y \partial x} \equiv 0$	. . . only if incompressible: $\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = -\frac{\partial^2 \phi}{\partial x \partial x} - \frac{\partial^2 \phi}{\partial y \partial y} = 0$

and

$$V_r = -\frac{\partial \phi}{\partial r} \qquad \text{and} \qquad V_\theta = -\frac{1}{r}\frac{\partial \phi}{\partial \theta} \tag{6.33}$$

In Section 5.2 we showed that the stream function ψ is constant along any streamline. For $\psi = \text{constant}$, $d\psi = 0$ and

$$d\psi = \frac{\partial \psi}{\partial x}dx + \frac{\partial \psi}{\partial y}dy = 0$$

The slope of a streamline—a line of constant ψ—is given by

$$\left.\frac{dy}{dx}\right)_\psi = -\frac{\partial \psi/dx}{\partial x/\partial y} = -\frac{-v}{u} = \frac{v}{u} \tag{6.34}$$

Along a line of constant ϕ, $d\phi = 0$ and

$$d\phi = \frac{\partial \phi}{\partial x}dx + \frac{\partial \phi}{\partial y}dy = 0$$

Consequently, the slope of a potential line — a line of constant ϕ — is given by

$$\left.\frac{dy}{dx}\right)_\phi = -\frac{\partial \phi/\partial x}{\partial \phi/\partial y} = -\frac{u}{v} \tag{6.35}$$

(The last equality of Eq. 6.35 follows from use of Eq. 6.29.)

Comparing Eqs. 6.34 and 6.35, we see that the slope of a constant ψ line at any point is the negative reciprocal of the slope of the constant ϕ line at that point; this means that *lines of constant ψ and constant ϕ are orthogonal*. This property of potential lines and streamlines is useful in graphical analyses of flow fields.

Example 6.10 VELOCITY POTENTIAL

Consider the flow field given by $\psi = ax^2 - ay^2$, where $a = 3\ \text{s}^{-1}$. Show that the flow is irrotational. Determine the velocity potential for this flow.

Given: Incompressible flow field with $\psi = ax^2 - ay^2$, where $a = 3\ \text{s}^{-1}$.

Find: (a) Whether or not the flow is irrotational.
(b) The velocity potential for this flow.

Solution: If the flow is irrotational, $\nabla^2\psi = 0$. Checking for the given flow,

$$\nabla^2\psi = \frac{\partial^2}{\partial x^2}(ax^2 - ay^2) + \frac{\partial^2}{\partial y^2}(ax^2 - ay^2) = 2a - 2a = 0$$

so that the flow *is* irrotational. As an alternative proof, we can compute the fluid particle rotation (in the xy plane, the only component of rotation is ω_z):

$$2\omega_z = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \qquad \text{and} \qquad u = \frac{\partial \psi}{\partial y} \quad v = -\frac{\partial \psi}{\partial x}$$

then

$$u = \frac{\partial}{\partial y}(ax^2 - ay^2) = -2ay \qquad \text{and} \qquad v = -\frac{\partial}{\partial x}(ax^2 - ay^2) = -2ax$$

so

$$2\omega_z = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = \frac{\partial}{\partial x}(-2ax) - \frac{\partial}{\partial y}(-2ay) = -2a + 2a = 0 \longleftarrow 2\omega_z$$

Once again, we conclude that the flow is irrotational. Because it is irrotational, ϕ must exist, and

$$u = -\frac{\partial \phi}{\partial x} \qquad \text{and} \qquad v = -\frac{\partial \phi}{\partial y}$$

Consequently, $u = -\dfrac{\partial \phi}{\partial x} = -2ay$ and $\dfrac{\partial \phi}{\partial x} = 2ay$. Integrating with respect to x gives $\phi = 2axy + f(y)$, where $f(y)$ is an arbitrary function of y. Then

$$v = -2ax = -\frac{\partial \phi}{\partial y} = -\frac{\partial}{\partial x}[2axy + f(y)]$$

Therefore, $-2ax = -2ax - \dfrac{\partial f(y)}{\partial y} = -2ax - \dfrac{df}{dy}$, so $\dfrac{df}{dy} = 0$ and $f = \text{constant}$. Thus

$$\phi = 2axy + \text{constant} \longleftarrow \phi$$

We also can show that lines of constant ψ and constant ϕ are orthogonal.

$$\psi = ax^2 - ay^2 \qquad \text{and} \qquad \phi = 2axy$$

For $\psi = \text{constant}$, $d\psi = 0 = 2axdx - 2aydy$; hence $\left.\dfrac{dy}{dx}\right)_{\psi = c} = \dfrac{x}{y}$

For $\phi = \text{constant}$, $d\phi = 0 = 2aydx + 2axdy$; hence $\left.\dfrac{dy}{dx}\right)_{\phi = c} = -\dfrac{y}{x}$

The slopes of lines of constant ϕ and constant ψ are negative reciprocals. Therefore lines of constant ϕ are orthogonal to lines of constant ψ.

This problem illustrates the relations among the stream function, velocity potential, and velocity field.

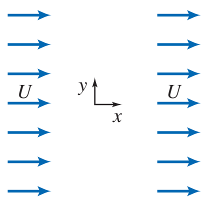
The stream function ψ and velocity potential ϕ are shown in the *Excel* workbook. By entering the equations for ψ and ϕ, other fields can be plotted.

Elementary Plane Flows

The ψ and ϕ functions for five elementary two-dimensional flows—a uniform flow, a source, a sink, a vortex, and a doublet—are summarized in Table 6.2. The ψ and ϕ functions can be obtained from the velocity field for each elementary flow. (We saw in Example 6.10 that we can obtain ϕ from u and v.)

Table 6.2

Elementary Plane Flows

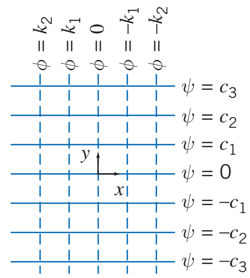


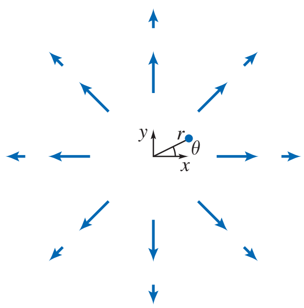
Uniform Flow (positive x direction)

$u = U \qquad \psi = Uy$

$v = 0 \qquad \phi = -Ux$

$\Gamma = 0$ around any closed curve





Source Flow (from origin)

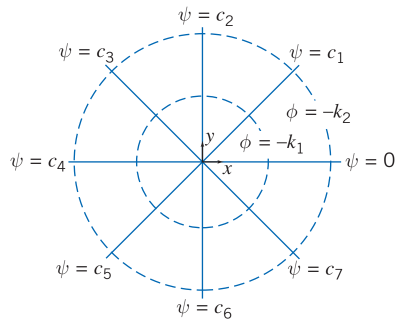
$$V_r = \frac{q}{2\pi r} \qquad \psi = \frac{q}{2\pi}\theta$$

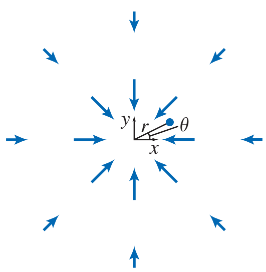
$$V_\theta = 0 \qquad \phi = -\frac{q}{2\pi}\ln r$$

Origin is singular point

q is volume flow rate per unit depth

$\Gamma = 0$ around any closed curve





Sink Flow (toward origin)

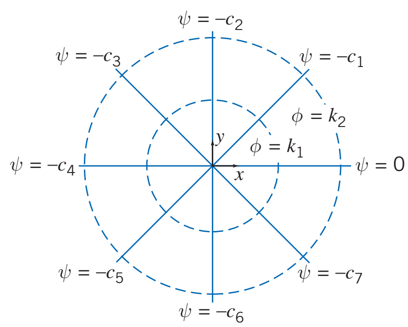
$$V_r = -\frac{q}{2\pi r} \qquad \psi = -\frac{q}{2\pi}\theta$$

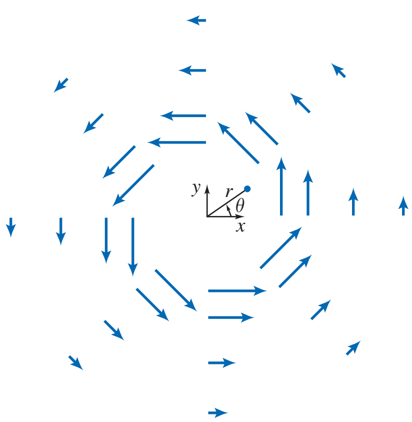
$$V_\theta = 0 \qquad \phi = \frac{q}{2\pi}\ln r$$

Origin is singular point

q is volume flow rate per unit depth

$\Gamma = 0$ around any closed curve





Irrotational Vortex (counterclockwise, center at origin)

$$V_r = 0 \qquad \psi = -\frac{K}{2\pi}\ln r$$

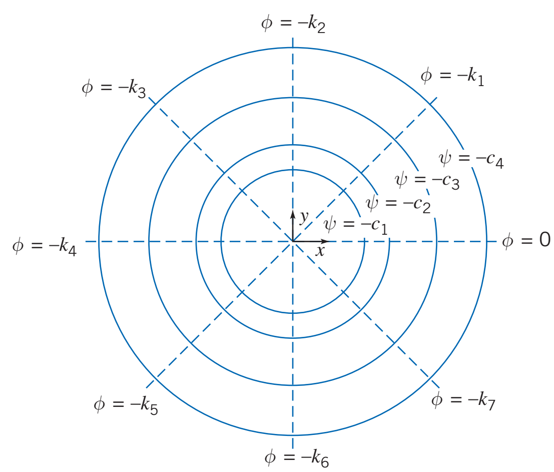
$$V_\theta = \frac{K}{2\pi r} \qquad \phi = -\frac{K}{2\pi}\theta$$

Origin is singular point

K is strength of the vortex

$\Gamma = K$ around any closed curve enclosing origin

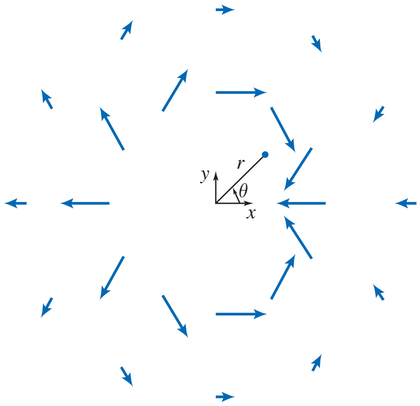
$\Gamma = 0$ around any closed curve not enclosing origin



(*Continued*)

Table 6.2

Elementary Plane Flows (*Continued*)

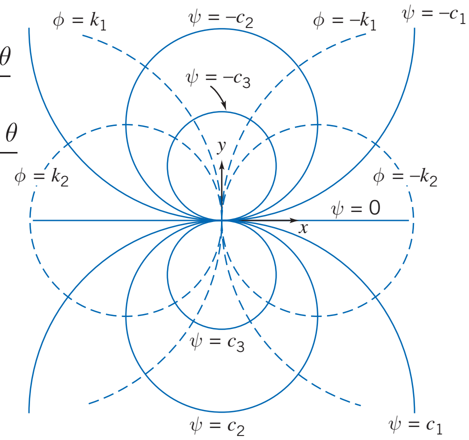


Doublet (center at origin)

$$V_r = -\frac{\Lambda}{r^2}\cos\theta \quad \psi = -\frac{\Lambda \sin\theta}{r}$$

$$V_\theta = -\frac{\Lambda}{r^2}\sin\theta \quad \phi = -\frac{\Lambda \cos\theta}{r}$$

Origin is singular point
Λ is strength of the doublet
$\Gamma =$ around any closed curve



A *uniform flow* of constant velocity parallel to the x axis satisfies the continuity equation and the irrotationality condition identically. In Table 6.2 we have shown the ψ and ϕ functions for a uniform flow in the positive x direction.

For a uniform flow of constant magnitude V, inclined at angle α to the x axis,

$$\psi = (V\cos\alpha)y - (V\sin\alpha)x$$
$$\phi = -(V\sin\alpha)y - (V\cos\alpha)x$$

A simple *source* is a flow pattern in the xy plane in which flow is radially outward from the z axis and symmetrical in all directions. The strength, q, of the source is the volume flow rate per unit depth. At any radius, r, from a source, the tangential velocity, V_θ, is zero; the radial velocity, V_r, is the volume flow rate per unit depth, q, divided by the flow area per unit depth, $2\pi r$. Thus $V_r = q/2\pi r$ for a source. Knowing V_r and V_θ, obtaining ψ and ϕ from Eqs. 5.8 and 6.33, respectively, is straightforward.

In a simple *sink*, flow is radially inward; a sink is a negative source. The ψ and ϕ functions for a sink shown in Table 6.2 are the negatives of the corresponding functions for a source flow.

The origin of either a sink or a source is a singular point, since the radial velocity approaches infinity as the radius approaches zero. Thus, while an actual flow may resemble a source or a sink for some values of r, sources and sinks have no exact physical counterparts. The primary value of the concept of sources and sinks is that, when combined with other elementary flows, they produce flow patterns that adequately represent realistic flows.

A flow pattern in which the streamlines are concentric circles is a vortex; in a *free (irrotational) vortex*, fluid particles do not rotate as they translate in circular paths around the vortex center. There are a number of ways of obtaining the velocity field, for example, by combining the equation of motion (Euler's equation) and the Bernoulli equation to eliminate the pressure. Here, though, for circular streamlines, we have $V_r = 0$ and $V_\theta = f(\theta)$ only. We also have previously introduced the condition of irrotationality in cylindrical coordinates,

$$\frac{1}{r}\frac{\partial rV_\theta}{\partial r} - \frac{1}{r}\frac{\partial V_r}{\partial \theta} = 0 \tag{6.24}$$

Hence, using the known forms of V_r and V_θ, we obtain

$$\frac{1}{r}\frac{d(rV_\theta)}{dr} = 0$$

Integrating this equation gives

$$V_\theta r = \text{constant}$$

The strength, K, of the vortex is defined as $K = 2\pi r V_\theta$; the dimensions of K are L^2/t (volume flow rate per unit depth). Once again, knowing V_r and V_θ, obtaining ψ and ϕ from Eqs. 5.8 and 6.33, respectively, is straightforward. The irrotational vortex is a reasonable approximation to the flow field in a tornado (except in the region of the origin; the origin is a singular point).

The final "elementary" flow listed in Table 6.2 is the *doublet* of strength Λ. This flow is produced mathematically by allowing a source and a sink of numerically equal strengths to merge. In the limit, as the distance, δs, between them approaches zero, their strengths increase so the product $q\delta s/2\pi$ tends to a finite value, Λ, which is termed the strength of the doublet.

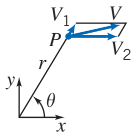
Superposition of Elementary Plane Flows

We saw earlier that both ϕ and ψ satisfy Laplace's equation for flow that is both incompressible and irrotational. Since Laplace's equation is a linear, homogeneous partial differential equation, solutions may be superposed (added together) to develop more complex and interesting patterns of flow. Thus if ψ_1 and ψ_2 satisfy Laplace's equation, then so does $\psi_3 = \psi_1 + \psi_2$. The elementary plane flows are the building blocks in this superposition process. There is one note of caution: While Laplace's equation for the stream function, and the stream function-velocity field equations (Eq. 5.3) are linear, the Bernoulli equation is not; hence, in the superposition process we will have $\psi_3 = \psi_1 + \psi_2$, $u_3 = u_1 + u_2$, and $v_3 = v_1 + v_2$, but $p_3 \neq p_1 + p_2$! We must use the Bernoulli equation, which is nonlinear in V, to find p_3.

We can add together elementary flows to try and generate recognizable flow patterns. The simplest superposition approach is called the *direct* method, in which we try different combinations of elementary flows and see what kinds of flow patterns are produced. This sounds like a random process, but with a little experience it becomes a quite logical process. For example, look at some of the classic examples listed in Table 6.3. The source and uniform flow combination makes sense—we would intuitively expect a source to partially push its way upstream, and to divert the flow around it. The source, sink, and uniform flow (generating what is called a Rankine body) is also not surprising—the entire flow out of the source makes its way into the sink, leading to a closed streamline. *Any streamline can be interpreted as a solid surface because there is no flow across it;* we can therefore pretend that this closed streamline represents a solid. We could easily generalize this source-sink approach to any number of sources and sinks distributed along the x axis, and as long as the sum of the source and sink strengths added up to zero, we would generate a closed streamline body shape. The doublet-uniform flow (with or without a vortex) generates a very interesting result: flow over a cylinder (with or without circulation)! We first saw the flow without circulation in Fig. 2.12*a*. The flow with a clockwise vortex produces a top-to-bottom asymmetry. This is because in the region above the cylinder the velocities due to the uniform flow and vortex are in the same overall direction and lead to a high velocity; below the cylinder they are in opposite directions and therefore lead to a low velocity. As we have learned, whenever velocities are high, streamlines will be close together, and vice versa—explaining the pattern shown. More importantly, from the Bernoulli equation we know that whenever the velocity is high the pressure will be low, and vice versa—hence, we can anticipate that the cylinder with circulation will experience a net upward force (lift) due to pressure. This approach, of looking at

Table 6.3
Superposition of Elementary Plane Flows

Source and Uniform Flow (flow past a half-body)

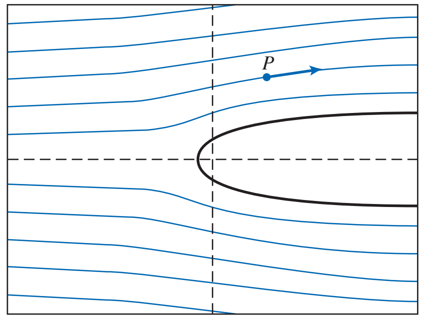


$$\psi = \psi_{so} + \psi_{uf} = \psi_1 + \psi_2 = \frac{q}{2\pi}\theta + Uy$$

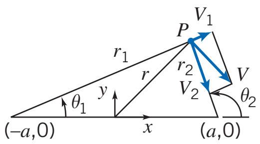
$$\psi = \frac{q}{2\pi}\theta + Ur\sin\theta$$

$$\phi = \phi_{so} + \phi_{uf} = \phi_1 + \phi_2 = -\frac{q}{2\pi}\ln r - Ux$$

$$\phi = -\frac{q}{2\pi}\ln r - Ur\cos\theta$$



Source and Sink (equal strength, separation distance on x axis = 2a)

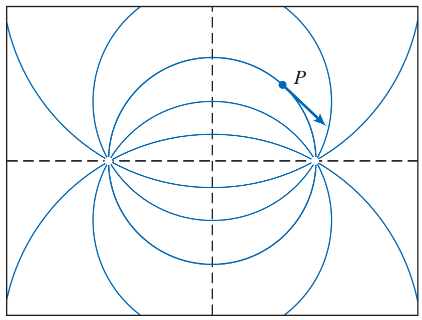


$$\psi = \psi_{so} + \psi_{si} = \psi_1 + \psi_2 = \frac{q}{2\pi}\theta_1 - \frac{q}{2\pi}\theta_2$$

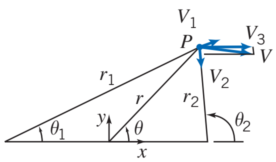
$$\psi = \frac{q}{2\pi}(\theta_1 - \theta_2)$$

$$\phi = \phi_{so} + \phi_{si} = \phi_1 + \phi_2 = -\frac{q}{2\pi}\ln r_1 + \frac{q}{2\pi}\ln r_2$$

$$\phi = \frac{q}{2\pi}\ln\frac{r_2}{r_1}$$



Source, Sink, and Uniform Flow (flow past a Rankine body)

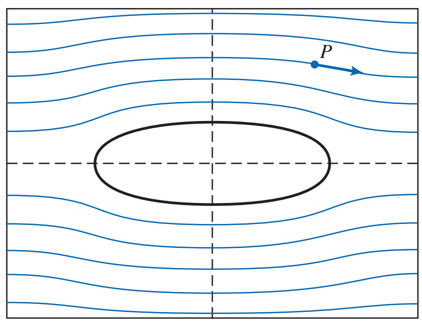


$$\psi = \psi_{so} + \psi_{si} + \psi_{uf} = \psi_1 + \psi_2 + \psi_3$$
$$= \frac{q}{2\pi}\theta_1 - \frac{q}{2\pi}\theta_2 + Uy$$

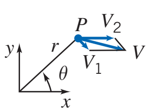
$$\psi = \frac{q}{2\pi}(\theta_1 - \theta_2) + Ur\sin\theta$$

$$\phi = \phi_{so} + \phi_{si} + \phi_{uf} = \phi_1 + \phi_2 + \phi_3$$
$$= -\frac{q}{2\pi}\ln r_1 + \frac{q}{2\pi}\ln r_2 - Ux$$

$$\phi = \frac{q}{2\pi}\ln\frac{r_2}{r_1} - Ur\cos\theta$$



Vortex (clockwise) and Uniform Flow



$$\psi = \psi_v + \psi_{uf} = \psi_1 + \psi_2 = \frac{K}{2\pi}\ln r + Uy$$

$$\psi = \frac{K}{2\pi}\ln r + Ur\sin\theta$$

$$\phi = \phi_v + \phi_{uf} = \phi_1 + \phi_2 = \frac{K}{2\pi}\theta - Ux$$

$$\phi = \frac{K}{2\pi}\theta - Ur\cos\theta$$

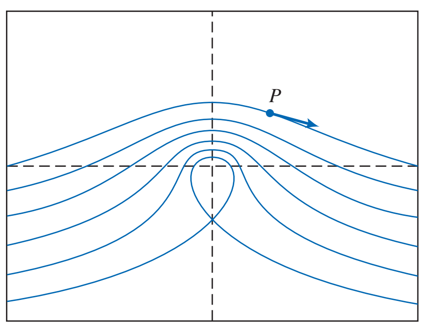
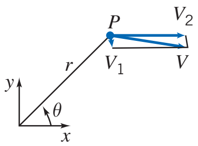


Table 6.3
Superposition of Elementary Plane Flows (*Continued*)

Doublet and Uniform Flow (flow past a cylinder)



$$\psi = \psi_d + \psi_{uf} = \psi_1 + \psi_2 = -\frac{\Lambda \sin\theta}{r} + Uy$$

$$= -\frac{\Lambda \sin\theta}{r} + Ur\sin\theta$$

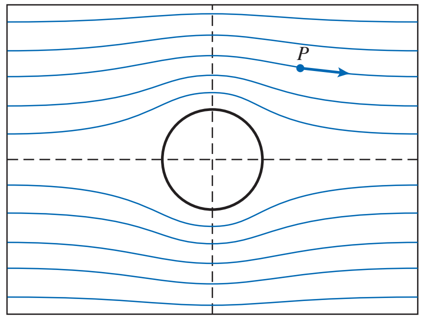
$$\psi = U\left(r - \frac{\Lambda}{Ur}\right)\sin\theta$$

$$\psi = Ur\left(1 - \frac{a^2}{r^2}\right)\sin\theta \quad a = \sqrt{\frac{\Lambda}{U}}$$

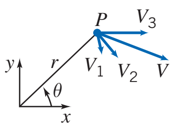
$$\phi = \phi_d + \phi_{uf} = \phi_1 + \phi_2 = -\frac{\Lambda\cos\theta}{r} - Ux$$

$$= -\frac{\Lambda\cos\theta}{r} - Ur\cos\theta$$

$$\phi = -U\left(r + \frac{\Lambda}{Ur}\right)\cos\theta = -Ur\left(1 + \frac{a^2}{r^2}\right)\cos\theta$$



Doublet, Vortex (clockwise), and Uniform Flow (flow past a cylinder with circulation)



$$\psi = \psi_d + \psi_v + \psi_{uf} = \psi_1 + \psi_2 + \psi_3$$

$$= -\frac{\Lambda\sin\theta}{r} + \frac{K}{2\pi}\ln r + Uy$$

$$\psi = -\frac{\Lambda\sin\theta}{r} + \frac{K}{2\pi}\ln r + Ur\sin\theta$$

$$\psi = Ur\left(1 - \frac{a^2}{r^2}\right)\sin\theta + \frac{K}{2\pi}\ln r$$

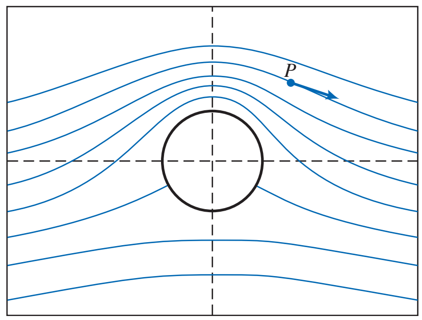
$$\phi = \phi_d + \phi_v + \phi_{uf} = \phi_1 + \phi_2 + \phi_3$$

$$= -\frac{\Lambda\cos\theta}{r} + \frac{K}{2\pi}\theta - Ux$$

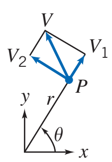
$$a = \sqrt{\frac{\Lambda}{U}}; \; K < 4\pi a U$$

$$\phi = -\frac{\Lambda\cos\theta}{r} + \frac{K}{2\pi}\theta - Ur\cos\theta$$

$$\phi = -Ur\left(1 + \frac{a^2}{r^2}\right)\cos\theta + \frac{K}{2\pi}\theta$$

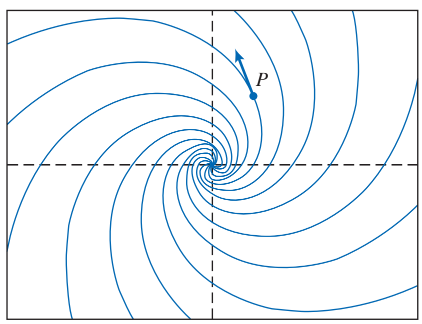


Source and Vortex (spiral vortex)



$$\psi = \psi_{so} + \psi_v = \psi_1 + \psi_2 = \frac{q}{2\pi}\theta - \frac{K}{2\pi}\ln r$$

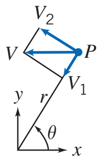
$$\phi = \phi_{so} + \phi_v = \phi_1 + \phi_2 = -\frac{q}{2\pi}\ln r - \frac{K}{2\pi}\theta$$



(*Continued*)

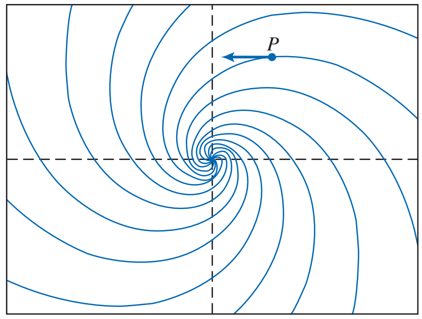
Table 6.3
Superposition of Elementary Plane Flows (*Continued*)

Sink and Vortex

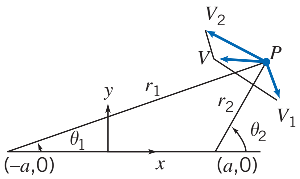


$$\psi = \psi_{si} + \psi_v = \psi_1 + \psi_2 = -\frac{q}{2\pi}\theta - \frac{K}{2\pi}\ln r$$

$$\phi = \phi_{si} + \phi_v = \phi_1 + \phi_2 = \frac{q}{2\pi}\ln r - \frac{K}{2\pi}\theta$$



Vortex Pair (equal strength, opposite rotation, separation distance on x axis = 2a)

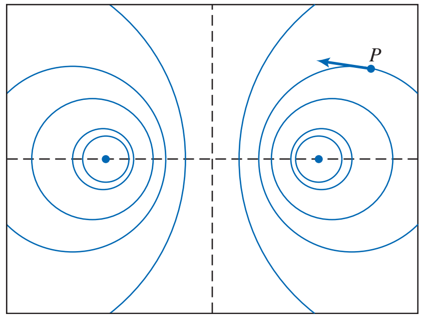


$$\psi = \psi_{v1} + \psi_{v2} = \psi_1 + \psi_2 = -\frac{K}{2\pi}\ln r_1 + \frac{K}{2\pi}\ln r_2$$

$$\psi = \frac{K}{2\pi}\ln\frac{r_2}{r_1}$$

$$\phi = \phi_{v1} + \phi_{v2} = \phi_1 + \phi_2 = -\frac{K}{2\pi}\theta_1 + \frac{K}{2\pi}\theta_2$$

$$\phi = \frac{K}{2\pi}(\theta_2 - \theta_1)$$



streamline patterns to see where we have regions of high or low velocity and hence low or high pressure, is very useful. We will examine these last two flows in Examples 6.11 and 6.12. The last example in Table 6.3, the vortex pair, hints at a way to create flows that simulate the presence of a wall or walls: for the y axis to be a streamline (and thus a wall), simply make sure that any objects (e.g., a source, a vortex) in the positive x quadrants have mirror-image objects in the negative x quadrants; the y axis will thus be a line of symmetry. For a flow pattern in a 90° corner, we need to place objects so that we have symmetry with respect to both the x and y axes. For flow in a corner whose angle is a fraction of 90° (e.g., 30°), we need to place objects in a radially symmetric fashion.

Because Laplace's equation appears in many engineering and physics applications, it has been extensively studied. We saw in Example 5.12 that Laplace's equation is sometimes amenable to a fairly simple numerical solution using a spreadsheet. For analytic solutions, one approach is to use conformal mapping with complex notation. It turns out that *any* continuous complex function $f(z)$ (where $z = x + iy$, and $i = \sqrt{-1}$) is a solution of Laplace's equation, and can therefore represent both ϕ and ψ. With this approach many elegant mathematical results have been derived [7–10]. We mention only two: the circle theorem, which enables any given flow [e.g., from a source at some point (a, b)] to be easily transformed to allow for the presence of a cylinder at the origin; and the Schwarz-Christoffel theorem, which enables a given flow to be transformed to allow for the presence of virtually unlimited stepwise linear boundaries (e.g., the presence on the x axis of the silhouette of a building).

Much of this analytical work was done centuries ago, when it was called "hydrodynamics" instead of potential theory. A list of famous contributors includes Bernoulli, Lagrange, d'Alembert, Cauchy, Rankine, and Euler [11]. As we discussed in Section 2.6, the theory immediately ran into difficulties: In an ideal fluid flow no body experiences drag—the d'Alembert paradox of 1752—a result completely counter to experience. Prandtl, in 1904, resolved this discrepancy by describing how real flows may be essentially inviscid almost everywhere, but there is always a "boundary layer" adjacent to the body. In this layer significant viscous effects occur, and the no-slip condition is satisfied (in potential flow theory the no-slip condition is not satisfied). Development of this concept, and the Wright brothers' historic first human flight, led to rapid developments in aeronautics starting in the 1900s. We will study boundary layers in detail in Chapter 9, where we will see that their existence leads to drag on bodies, and also affects the lift of bodies.

An alternative superposition approach is the *inverse* method in which distributions of objects such as sources, sinks, and vortices are used to model a body [12]. It is called inverse because the body shape is deduced based on a desired pressure distribution. Both the direct and inverse methods, including three-dimensional space, are today mostly analyzed using computer applications such as *Fluent* [13] and *STAR-CD* [14].

Example 6.11 FLOW OVER A CYLINDER: SUPERPOSITION OF DOUBLET AND UNIFORM FLOW

For two-dimensional, incompressible, irrotational flow, the superposition of a doublet and a uniform flow represents flow around a circular cylinder. Obtain the stream function and velocity potential for this flow pattern. Find the velocity field, locate the stagnation points and the cylinder surface, and obtain the surface pressure distribution. Integrate the pressure distribution to obtain the drag and lift forces on the circular cylinder.

Given: Two-dimensional, incompressible, irrotational flow formed from superposition of a doublet and a uniform flow.

Find:
(a) Stream function and velocity potential.
(b) Velocity field.
(c) Stagnation points.
(d) Cylinder surface.
(e) Surface pressure distribution.
(f) Drag force on the circular cylinder.
(g) Lift force on the circular cylinder.

Solution: Stream functions may be added because the flow field is incompressible and irrotational. Thus from Table 6.2, the stream function for the combination is

$$\psi = \psi_d + \psi_{uf} = -\frac{\Lambda \sin\theta}{r} + Ur\sin\theta \longleftarrow \psi$$

The velocity potential is

$$\phi = \phi_d + \phi_{uf} = -\frac{\Lambda \cos\theta}{r} - Ur\cos\theta \longleftarrow \phi$$

The corresponding velocity components are obtained using Eqs. 6.30 as

$$V_r = -\frac{\partial\phi}{\partial r} = -\frac{\Lambda\cos\theta}{r^2} + U\cos\theta$$

$$V_\theta = -\frac{1}{r}\frac{\partial\phi}{\partial\theta} = -\frac{\Lambda\sin\theta}{r^2} - U\sin\theta$$

The velocity field is

$$\vec{V} = V_r\hat{e}_r + V_\theta\hat{e}_\theta = \left(-\frac{\Lambda\cos\theta}{r^2} + U\cos\theta\right)\hat{e}_r + \left(-\frac{\Lambda\sin\theta}{r^2} - U\sin\theta\right)\hat{e}_\theta \longleftarrow \vec{V}$$

Stagnation points are where $\vec{V} = V_r\hat{e}_r + V_\theta\hat{e}_\theta = 0$

$$V_r = -\frac{\Lambda\cos\theta}{r^2} + U\cos\theta = \cos\theta\left(U - \frac{\Lambda}{r^2}\right)$$

Thus $V_r = 0$ when $r = \sqrt{\frac{\Lambda}{U}} = a$. Also,

$$V_\theta = -\frac{\Lambda\sin\theta}{r^2} - U\sin\theta = -\sin\theta\left(U + \frac{\Lambda}{r^2}\right)$$

Thus $V_\theta = 0$ when $\theta = 0, \pi$.
Stagnation points are $(r,\theta) = (a,0), (a,\pi)$. ⟵ Stagnation points

Note that $V_r = 0$ along $r = a$, so this represents flow around a circular cylinder, as shown in Table 6.3. Flow is irrotational, so the Bernoulli equation may be applied between any two points. Applying the equation between a point far upstream and a point on the surface of the cylinder (neglecting elevation differences), we obtain

$$\frac{p_\infty}{\rho} + \frac{U^2}{2} = \frac{p}{\rho} + \frac{V^2}{2}$$

Thus,

$$p - p_\infty = \frac{1}{2}\rho(U^2 - V^2)$$

Along the surface, $r = a$, and

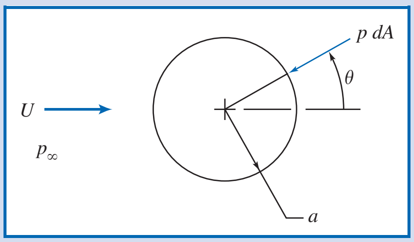
$$V^2 = V_\theta^2 = \left(-\frac{\Lambda}{a^2} - U\right)^2 \sin^2\theta = 4U^2\sin^2\theta$$

since $\Lambda = Ua^2$. Substituting yields

$$p - p_\infty = \frac{1}{2}\rho(U^2 - 4U^2\sin^2\theta) = \frac{1}{2}\rho U^2(1 - 4\sin^2\theta)$$

or

$$\frac{p - p_\infty}{\frac{1}{2}\rho U^2} = 1 - 4\sin^2\theta \longleftarrow \text{Pressure distribution}$$



Drag is the force component parallel to the freestream flow direction. The drag force is given by

$$F_D = \int_A -p\,dA\cos\theta = \int_0^{2\pi} -pa\,d\theta\,b\cos\theta$$

since $dA = a\,d\theta\,b$, where b is the length of the cylinder normal to the diagram.

Substituting $p = p_\infty + \frac{1}{2}\rho U^2(1 - 4\sin^2\theta)$,

$$F_D = \int_0^{2\pi} -p_\infty ab \cos\theta \, d\theta + \int_0^{2\pi} -\frac{1}{2}\rho U^2 (1 - 4\sin^2\theta) ab \cos\theta \, d\theta$$

$$= -p_\infty \, ab \sin\theta \Big]_0^{2\pi} - \frac{1}{2}\rho U^2 ab \sin\theta \Big]_0^{2\pi} + \frac{1}{2} p U^2 ab \frac{4}{3} \sin^3\theta \Big]_0^{2\pi}$$

$$F_D = 0 \longleftarrow F_D$$

Lift is the force component normal to the freestream flow direction. (By convention, positive lift is an upward force.) The lift force is given by

$$F_L = \int_A p \, dA(-\sin\theta) = -\int_0^{2\pi} pa \, d\theta \, b \sin\theta$$

Substituting for p gives

$$F_L = -\int_0^{2\pi} p_\infty ab \sin\theta \, d\theta - \int_0^{2\pi} \frac{1}{2}\rho U^2 (1 - 4\sin^2\theta) ab \sin\theta \, d\theta$$

$$= p_\infty a \, b \cos\theta \Big]_0^{2\pi} + \frac{1}{2}\rho U^2 ab \cos\theta \Big]_0^{2\pi} + \frac{1}{2}\rho U^2 ab \left[\frac{4\cos^3\theta}{3} - 4\cos\theta \right]_0^{2\pi}$$

$$F_L = 0 \longleftarrow F_L$$

This problem illustrates:
- ✓ How elementary plane flows can be combined to generate interesting and useful flow patterns.
- ✓ d'Alembert's paradox, that potential flows over a body do not generate drag.

The stream function and pressure distribution are plotted in the *Excel* workbook.

Example 6.12 FLOW OVER A CYLINDER WITH CIRCULATION: SUPERPOSITION OF DOUBLET, UNIFORM FLOW, AND CLOCKWISE FREE VORTEX

For two-dimensional, incompressible, irrotational flow, the superposition of a doublet, a uniform flow, and a free vortex represents the flow around a circular cylinder with circulation. Obtain the stream function and velocity potential for this flow pattern, using a clockwise free vortex. Find the velocity field, locate the stagnation points and the cylinder surface, and obtain the surface pressure distribution. Integrate the pressure distribution to obtain the drag and lift forces on the circular cylinder. Relate the lift force on the cylinder to the circulation of the free vortex.

Given: Two-dimensional, incompressible, irrotational flow formed from superposition of a doublet, a uniform flow, and a clockwise free vortex.

Find:
(a) Stream function and velocity potential.
(b) Velocity field.
(c) Stagnation points.
(d) Cylinder surface.
(e) Surface pressure distribution.
(f) Drag force on the circular cylinder.
(g) Lift force on the circular cylinder.
(h) Lift force in terms of circulation of the free vortex.

Solution:

Stream functions may be added because the flow field is incompressible and irrotational. From Table 6.2, the stream function and velocity potential for a clockwise free vortex are

$$\psi_{fv} = \frac{K}{2\pi}\ln r \qquad \phi_{fv} = \frac{K}{2\pi}\theta$$

Using the results of Example 6.11, the stream function for the combination is

$$\psi = \psi_d + \psi_{uf} + \psi_{fv}$$

$$\psi = -\frac{\Lambda \sin\theta}{r} + Ur\sin\theta + \frac{K}{2\pi}\ln r$$

ψ

The velocity potential for the combination is

$$\phi = \phi_d + \phi_{uf} + \phi_{fv}$$

$$\phi = -\frac{\Lambda \cos\theta}{r} - Ur\cos\theta + \frac{K}{2\pi}\theta$$

ϕ

The corresponding velocity components are obtained using Eqs. 6.30 as

$$V_r = -\frac{\partial \phi}{\partial r} = -\frac{\Lambda \cos\theta}{r^2} + U\cos\theta \tag{1}$$

$$V_\theta = -\frac{1}{r}\frac{\partial \phi}{\partial \theta} = -\frac{\Lambda \sin\theta}{r^2} - U\sin\theta - \frac{K}{2\pi r} \tag{2}$$

The velocity field is

$$\vec{V} = V_r\,\hat{e}_r + V_\theta\,\hat{e}_\theta$$

$$\vec{V} = \left(-\frac{\Lambda\cos\theta}{r^2} + U\cos\theta\right)\hat{e}_r + \left(-\frac{\Lambda\sin\theta}{r} - U\sin\theta - \frac{K}{2\pi r}\right)\hat{e}_\theta$$

$\vec{V}$

Stagnation points are located where $\vec{V} = V_r\,\hat{e}_r + V_\theta\,\hat{e}_\theta = 0$. From Eq. 1,

$$V_r = -\frac{\Lambda\cos\theta}{r^2} + U\cos\theta = \cos\theta\left(U - \frac{\Lambda}{r^2}\right)$$

Thus $V_r = 0$ when $r = \sqrt{\Lambda/U} = a$

Cylinder surface

The stagnation points are located on $r = a$. Substituting into Eq. 2 with $r = a$,

$$V_\theta = -\frac{\Lambda\sin\theta}{a^2} - U\sin\theta - \frac{K}{2\pi a}$$

$$= -\frac{\Lambda\sin\theta}{\Lambda/U} - U\sin\theta - \frac{K}{2\pi a}$$

$$V_\theta = -2U\sin\theta - \frac{K}{2\pi a}$$

Thus $V_\theta = 0$ along $r = a$ when

$$\sin\theta = -\frac{K}{4\pi Ua} \qquad \text{or} \qquad \theta = \sin^{-1}\left[\frac{-K}{4\pi Ua}\right]$$

Stagnation points: $\quad r = a \qquad \theta = \sin^{-1}\left[\dfrac{-K}{4\pi Ua}\right]$

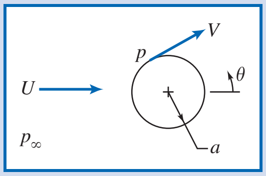
Stagnation points

As in Example 6.11, $V_r = 0$ along $r = a$, so this flow field once again represents flow around a circular cylinder, as shown in Table 6.3. For $K = 0$ the solution is identical to that of Example 6.11.

The presence of the free vortex ($K > 0$) moves the stagnation points below the center of the cylinder. Thus the free vortex alters the vertical symmetry of the flow field. The flow field has two stagnation points for a range of vortex strengths between $K = 0$ and $K = 4\pi Ua$.

A single stagnation point is located at $\theta = -\pi/2$ when $K = 4\pi Ua$.

Even with the free vortex present, the flow field is irrotational, so the Bernoulli equation may be applied between any two points. Applying the equation between a point far upstream and a point on the surface of the cylinder we obtain



$$\frac{p_\infty}{\rho} + \frac{U^2}{2} + gz = \frac{p}{\rho} + \frac{V^2}{2} + gz$$

Thus, neglecting elevation differences,

$$p - p_\infty = \frac{1}{2}\rho\,(U^2 - V^2) = \frac{1}{2}\rho U^2\left[1 - \left(\frac{U}{V}\right)^2\right]$$

Along the surface $r = a$ and $V_r = 0$, so

$$V^2 = V_\theta^2 = \left(-2U\sin\theta - \frac{K}{2\pi a}\right)^2$$

and

$$\left(\frac{V}{U}\right)^2 = 4\sin^2\theta + \frac{2K}{\pi Ua}\sin\theta + \frac{K^2}{4\pi^2 U^2 a^2}$$

Thus

$$p = p_\infty + \frac{1}{2}\rho U^2\left(1 - 4\sin^2\theta - \frac{2K}{\pi Ua}\sin\theta - \frac{K^2}{4\pi^2U^2a^2}\right) \longleftarrow p(\theta)$$

Drag is the force component parallel to the freestream flow direction. As in Example 6.11, the drag force is given by

$$F_D = \int_A -p\,dA\cos\theta = \int_0^{2\pi} -pa\,d\theta b\cos\theta$$

since $dA = a\,d\theta\,b$, where b is the length of the cylinder normal to the diagram.

Comparing pressure distributions, the free vortex contributes only to the terms containing the factor K. The contribution of these terms to the drag force is

$$\frac{F_{D_{fv}}}{\frac{1}{2}\rho U^2} = \int_0^{2\pi}\left(\frac{2K}{\pi Ua}\sin\theta + \frac{K^2}{4\pi^2U^2a^2}\right)ab\cos\theta\,d\theta \tag{3}$$

$$\frac{F_{D_{fv}}}{\frac{1}{2}\rho U^2} = \frac{2K}{\pi Ua}ab\left.\frac{\sin^2\theta}{2}\right]_0^{2\pi} + \frac{K^2}{4\pi^2U^2a^2}ab\sin\theta\Big]_0^{2\pi} = 0 \longleftarrow F_D$$

Lift is the force component normal to the freestream flow direction. (Upward force is defined as positive lift.) The lift force is given by

$$F_L = \int_A -p\,dA\sin\theta = \int_0^{2\pi} -pa\,d\theta\,b\sin\theta$$

Comparing pressure distributions, the free vortex contributes only to the terms containing the factor K. The contribution of these terms to the lift force is

$$\frac{F_{L_{fv}}}{\frac{1}{2}\rho U^2} = \int_0^{2\pi}\left(\frac{2K}{\pi Ua}\sin\theta + \frac{K^2}{4\pi^2 U^2 a^2}\right) ab\sin\theta\, d\theta$$

$$= \frac{2K}{\pi Ua}\int_0^{2\pi} ab\sin^2\theta d\theta + \frac{K^2}{4\pi^2 U^2 a^2}\int_0^{2\pi} ab\sin\theta\, d\theta$$

$$= \frac{2Kb}{\pi U}\left[\frac{\theta}{2} - \frac{\sin^2\theta}{4}\right]_0^{2\pi} - \frac{K^2 b}{4\pi^2 U^2 a}\cos\theta\Big]_0^{2\pi}$$

$$\frac{F_{L_{fv}}}{\frac{1}{2}\rho U^2} = \frac{2Kb}{\pi U}\left[\frac{2\pi}{2}\right] = \frac{2Kb}{U}$$

Then $F_{L_{fv}} = \rho UKb \longleftarrow$ F_L

The *circulation* is defined by Eq. 5.18 as

$$\Gamma \equiv \oint \vec{V}\cdot d\vec{s}$$

On the cylinder surface, $r = a$, and $\vec{V} = V_\theta\,\hat{e}_\theta$, so

$$\Gamma = \int_0^{2\pi}\left(-2U\sin\theta - \frac{K}{2\pi a}\right)\hat{e}_\theta \cdot a\, d\theta\,\hat{e}_\theta$$

$$= -\int_0^{2\pi} 2Ua\sin\theta\, d\theta - \int_0^{2\pi}\frac{K}{2\pi}\, d\theta$$

$\Gamma = -K \longleftarrow$ Circulation

Substituting into the expression for lift,

$$F_L = \rho UKb = \rho U(-\Gamma)b = -\rho U\,\Gamma b$$

or the lift force per unit length of cylinder is

$\frac{F_L}{b} = -\rho U\Gamma \longleftarrow$ $\frac{F_L}{b}$

This problem illustrates:

✓ Once again d'Alembert's paradox, that potential flows do not generate drag on a body.
✓ That the lift per unit length is $-\rho U\Gamma$. It turns out that this expression for lift is the same for *all* bodies in an ideal fluid flow, regardless of shape!

The stream function and pressure distribution are plotted in the *Excel* workbook.

6.8 Summary and Useful Equations

In this chapter we have:

✓ Derived Euler's equations in vector form and in rectangular, cylindrical, and streamline coordinates.
✓ Obtained Bernoulli's equation by integrating Euler's equation along a steady-flow streamline, and discussed its restrictions. We have also seen how for a steady, incompressible flow through a stream tube the first law of thermodynamics reduces to the Bernoulli equation if certain restrictions apply.

✓ Defined the static, dynamic, and stagnation (or total) pressures.
✓ Defined the energy and hydraulic grade lines.
✓ *Derived an unsteady flow Bernoulli equation, and discussed its restrictions.
✓ *Observed that for an irrotational flow that is steady and incompressible, the Bernoulli equation applies between *any* two points in the flow.
✓ *Defined the velocity potential ϕ and discussed its restrictions.

We have also explored in detail two-dimensional, incompressible, and irrotational flows, and learned that for these flows: the stream function ψ and the velocity potential ϕ satisfy Laplace's equation; ψ and ϕ can be derived from the velocity components, and vice versa, and the iso-lines of the stream function ψ and the velocity potential ϕ are orthogonal. We explored for such flows how to combine potential flows to generate various flow patterns, and how to determine the pressure distribution and lift and drag on, for example, a cylindrical shape.

Note: Most of the Useful Equations in the table below have a number of constraints or limitations—*be sure to refer to their page numbers for details*!

Useful Equations

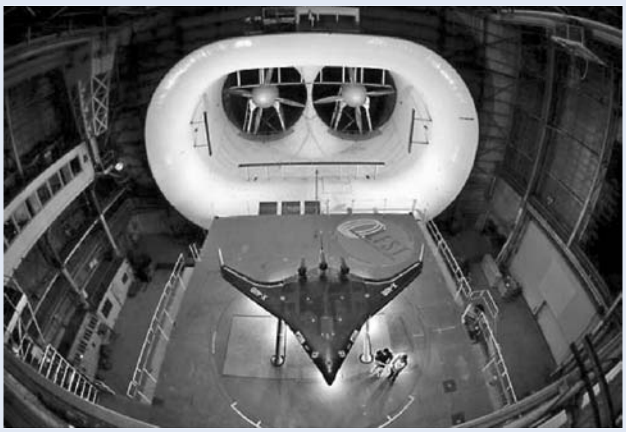
The Euler equation for incompressible, inviscid flow:	$\rho \dfrac{D\vec{V}}{Dt} = \rho\vec{g} - \nabla p$	**(6.1)**	Page 237
The Euler equation (rectangular coordinates):	$\rho\left(\dfrac{\partial u}{\partial t} + u\dfrac{\partial u}{\partial x} + v\dfrac{\partial u}{\partial y} + w\dfrac{\partial u}{\partial z}\right) = \rho g_x - \dfrac{\partial p}{\partial x}$	**(6.2a)**	Page 237
	$\rho\left(\dfrac{\partial v}{\partial t} + u\dfrac{\partial v}{\partial x} + v\dfrac{\partial v}{\partial y} + w\dfrac{\partial v}{\partial z}\right) = \rho g_y - \dfrac{\partial p}{\partial y}$	**(6.2b)**	
	$\rho\left(\dfrac{\partial w}{\partial t} + u\dfrac{\partial w}{\partial x} + v\dfrac{\partial w}{\partial y} + w\dfrac{\partial w}{\partial z}\right) = \rho g_z - \dfrac{\partial p}{\partial z}$	**(6.2c)**	
The Euler equation (cylindrical coordinates):	$\rho a_r = \rho\left(\dfrac{\partial V_r}{\partial t} + V_r\dfrac{\partial V_r}{\partial r} + \dfrac{V_\theta}{r}\dfrac{\partial V_r}{\partial \theta} + V_z\dfrac{\partial V_r}{\partial z} - \dfrac{V_\theta^2}{r}\right) = \rho g_r - \dfrac{\partial p}{\partial r}$	**(6.3a)**	Page 237
	$\rho a_\theta = \rho\left(\dfrac{\partial V_\theta}{\partial t} + V_r\dfrac{\partial V_\theta}{\partial r} + \dfrac{V_\theta}{r}\dfrac{\partial V_\theta}{\partial \theta} + V_z\dfrac{\partial V_\theta}{\partial z} + \dfrac{V_r V_\theta}{r}\right) = \rho g_\theta - \dfrac{1}{r}\dfrac{\partial p}{\partial \theta}$	**(6.3b)**	
	$\rho a_z = \rho\left(\dfrac{\partial V_z}{\partial t} + V_r\dfrac{\partial V_z}{\partial r} + \dfrac{V_\theta}{r}\dfrac{\partial V_z}{\partial \theta} + V_z\dfrac{\partial V_z}{\partial z}\right) = \rho g_z - \dfrac{\partial p}{\partial z}$	**(6.3c)**	
The Bernoulli equation (steady, incompressible, inviscid, along a streamline):	$\dfrac{p}{\rho} + \dfrac{V^2}{2} + gz = \text{constant}$	**(6.8)**	Page 242
Definition of total head of a flow:	$\dfrac{p}{\rho g} + \dfrac{V^2}{2g} + z = H$	**(6.16a)**	Page 258
Definition of energy grade line (EGL):	$EGL = \dfrac{p}{\rho g} + \dfrac{V^2}{2g} + z$	**(6.16b)**	Page 258
Definition of hydraulic grade line (HGL):	$HGL = \dfrac{p}{\rho g} + z$	**(6.16c)**	Page 258

*These topics apply to sections that may be omitted without loss of continuity in the text material.

Relation between EGL, HGL, and dynamic pressure:	$EGL - HGL = \frac{V^2}{2g}$	**(6.16d)**	Page 258
The unsteady Bernoulli equation (incompressible, inviscid, along a streamline):	$\frac{p_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{p_2}{\rho} + \frac{V_2^2}{2} + gz_2 + \int_1^2 \frac{\partial V}{\partial t}\, ds$	**(6.21)**	Page W-16
Definition of stream function (2D, incompressible flow):	$u = \frac{\partial \psi}{\partial y} \qquad v = -\frac{\partial \psi}{\partial x}$	**(5.4)**	Page 262
Definition of velocity potential (2D irrotational flow):	$u = -\frac{\partial \phi}{\partial x} \qquad v = -\frac{\partial \phi}{\partial y}$	**(6.29)**	Page 262
Definition of stream function (2D, incompressible flow, cylindrical coordinates):	$V_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta}$ and $V_\theta = -\frac{\partial \psi}{\partial r}$	**(5.8)**	Page 262
Definition of velocity potential (2D irrotational flow, cylindrical coordinates):	$V_r = -\frac{\partial \phi}{\partial r}$ and $V_\theta = -\frac{1}{r}\frac{\partial \phi}{\partial \theta}$	**(6.33)**	Page 263

Case Study

The Blended Wing-Body Aircraft



The X-48B prototype in the full-scale NASA tunnel. (Credit: Boeing/Bob Ferguson)

Boeing Phantom Works has partnered with NASA and the U.S. Air Force Research Laboratory to study an advanced-concept, fuel-efficient, blended wing-body. It is called a blended wing-body (BWB) because it looks more like a modified triangular-shaped wing than traditional aircraft, which essentially consist of a tube and wing with a tail. The concept of a BWB actually goes back to the 1940s, but developments in composite materials and fly-by-wire controls are making it more feasible. Researchers have tested a 6.3-m wingspan (8.5 percent scale) prototype of the X-48B, a BWB aircraft that could have military and commercial applications. The next step is for NASA to flight-test a scale-model variant called X-48C. The X-48C will be used to examine how engines mounted to the rear and above the body help to shield the ground from engine noise on takeoff and approach. It also features tail fins for additional noise shielding and for flight control.

The big difference between BWB aircraft and the traditional tube-and-wing aircraft, apart from the fact that the tube is absorbed into the wing shape, is that it does not have a tail. Traditional aircraft need a tail for stability and control; the BWB uses a number of different multiple-control surfaces and possibly tail fins to control the vehicle. There will be a number of advantages to the BWB if it proves feasible. Because the entire structure

generates lift, less power is needed for takeoff. Studies have also shown that BWB designs can fit into the 80-m (260-ft) envelope that is the current standard for airplane maneuver at airports. A BWB could carry up to 1000 people, making such a future U.S. product a challenge to Airbus's A380 and future stretched versions.

Apart from possible fuel savings of up to 30 percent due to improved streamlining, the interior of a commercial BWB airplane would be radically different from that of current airplanes. Passengers would enter a room like a movie theater rather than a cramped half-cylinder, there would be no windows (video screens would be connected to external cameras instead), and passengers would be seated in the large movie theater–like room (because seating is not only in the central core but also well out into the blended wings).

References

1. Shaw, R., "The Influence of Hole Dimensions on Static Pressure Measurements," *J. Fluid Mech., 7*, Part 4, April 1960, pp. 550–564.
2. Chue, S. H., "Pressure Probes for Fluid Measurement," *Progress in Aerospace Science, 16*, 2, 1975, pp. 147–223.
3. United Sensor Corporation, 3 Northern Blvd., Amherst, NH 03031.
4. Robertson, J. M., *Hydrodynamics in Theory and Application*. Englewood Cliffs, NJ: Prentice-Hall, 1965.
5. Streeter, V. L., *Fluid Dynamics*. New York: McGraw-Hill, 1948.
6. Vallentine, H. R., *Applied Hydrodynamics*. London: Butterworths, 1959.
7. Lamb, H., *Hydrodynamics*. New York: Dover, 1945.
8. Milne-Thomson, L. M., *Theoretical Hydrodynamics*, 4th ed. New York: Macmillan, 1960.
9. Karamcheti, K., *Principles of Ideal-Fluid Aerodynamics*. New York: Wiley, 1966.
10. Kirchhoff, R. H., *Potential Flows: Computer Graphic Solutions*. New York: Marcel Dekker, 1985.
11. Rouse, H., and S. Ince, *History of Hydraulics*. New York: Dover, 1957.
12. Kuethe, A. M., and C.-Y. Chow, *Foundations of Aerodynamics: Bases of Aerodynamic Design*, 4th ed. New York: Wiley, 1986.
13. *Fluent*. Fluent Incorporated, Centerra Resources Park, 10 Cavendish Court, Lebanon, NH 03766 (www.fluent.com).
14. *STAR-CD*. Adapco, 60 Broadhollow Road, Melville, NY 11747 (www.cd-adapco.com).

Problems

Euler's Equation

6.1 Consider the flow field with velocity given by $\vec{V} = [A(y^2 - x^2) - Bx]\hat{i} + [2Axy + By]\hat{j}$; $A = 1\ \text{ft}^{-1}\cdot\text{s}^{-1}$, $B = 1\ \text{ft}^{-1}\cdot\text{s}^{-1}$; the coordinates are measured in feet. The density is 2 slug/ft^3, and gravity acts in the negative y direction. Calculate the acceleration of a fluid particle and the pressure gradient at point $(x, y) = (1, 1)$.

6.2 An incompressible frictionless flow field is given by $\vec{V} = (Ax + By)\hat{i} + (Bx - Ay)\hat{j}$, where $A = 2\ \text{s}^{-1}$ and $B = 2\ \text{s}^{-1}$, and the coordinates are measured in meters. Find the magnitude and direction of the acceleration of a fluid particle at point $(x, y) = (2, 2)$. Find the pressure gradient at the same point, if $\vec{g} = -g\hat{j}$ and the fluid is water.

6.3 A horizontal flow of water is described by the velocity field $\vec{V} = (-Ax + Bt)\hat{i} + (Ay + Bt)\hat{j}$, where $A = 1\ \text{s}^{-1}$ and $B = 2\ \text{m/s}^2$, x and y are in meters, and t is in seconds. Find expressions for the local acceleration, the convective acceleration, and the total acceleration. Evaluate these at point $(1, 2)$ at $t = 5$ seconds. Evaluate ∇p at the same point and time.

6.4 A velocity field in a fluid with density of 1000 kg/m^3 is given by $\vec{V} = (-Ax + By)t\hat{i} + (Ay + Bx)t\hat{j}$, where $A = 2\ \text{s}^{-2}$ and $B = 1\ \text{s}^{-2}$, x and y are in meters, and t is in seconds. Body forces are negligible. Evaluate ∇p at point $(x, y) = (1, 1)$ at $t = 1$ s.

6.5 Consider the flow field with velocity given by $\vec{V} = [A(x^2 - y^2) - 3Bx]\hat{i} - [2Axy - 3By]\hat{j}$, where $A = 1\ \text{ft}^{-1}\cdot\text{s}^{-1}$, $B = 1\ \text{s}^{-1}$, and the coordinates are measured in feet. The density is 2 slug/ft^3 and gravity acts in the negative y direction. Determine the acceleration of a fluid particle and the pressure gradient at point $(x, y) = (1, 1)$.

6.6 The x component of velocity in an incompressible flow field is given by $u = Ax$, where $A = 2\ \text{s}^{-1}$ and the coordinates are measured in meters. The pressure at point $(x, y) = (0, 0)$ is $p_0 = 190$ kPa (gage). The density is $\rho = 1.50\ \text{kg/m}^3$ and the z axis is vertical. Evaluate the simplest possible y component

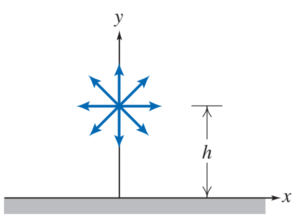
of velocity. Calculate the fluid acceleration and determine the pressure gradient at point $(x, y) = (2, 1)$. Find the pressure distribution along the positive x axis.

6.7 Consider the flow field with velocity given by $\vec{V} = Ax\sin(2\pi\omega t)\hat{i} - Ay\sin(2\pi\omega t)\hat{j}$, where $A = 2\text{ s}^{-1}$ and $\omega = 1\text{ s}^{-1}$. The fluid density is 2 kg/m^3. Find expressions for the local acceleration, the convective acceleration, and the total acceleration. Evaluate these at point (1, 1) at $t = 0$, 0.5, and 1 seconds. Evaluate ∇p at the same point and times.

6.8 The velocity field for a plane source located distance $h = 1$ m above an infinite wall aligned along the x axis is given by

$$\vec{V} = \frac{q}{2\pi[x^2 + (y-h)^2]}[x\hat{i} + (y-h)\hat{j}] + \frac{q}{2\pi[x^2 + (y+h)^2]}[x\hat{i} + (y+h)\hat{j}]$$

where $q = 2$ m^3/s/m. The fluid density is 1000 kg/m^3 and body forces are negligible. Derive expressions for the velocity and acceleration of a fluid particle that moves along the wall, and plot from $x = 0$ to $x = +10h$. Verify that the velocity and acceleration normal to the wall are zero. Plot the pressure gradient $\partial p/\partial x$ along the wall. Is the pressure gradient along the wall adverse (does it oppose fluid motion) or not?



P6.8

6.9 The velocity distribution in a two-dimensional steady flow field in the xy plane is $\vec{V} = (Ax - B)\hat{i} + (C - Ay)\hat{j}$, where $A = 2\text{ s}^{-1}$, $B = 5\text{ m}\cdot\text{s}^{-1}$, and $C = 3\text{ m}\cdot\text{s}^{-1}$; the coordinates are measured in meters, and the body force distribution is $\vec{g} = -g\hat{k}$. Does the velocity field represent the flow of an incompressible fluid? Find the stagnation point of the flow field. Obtain an expression for the pressure gradient in the flow field. Evaluate the difference in pressure between point $(x, y) = (1, 3)$ and the origin, if the density is 1.2 kg/m^3.

6.10 In a two-dimensional frictionless, incompressible ($\rho = 1500$ kg/m^3) flow, the velocity field in meters per second is given by $\vec{V} = (Ax + By)\hat{i} + (Bx - Ay)\hat{j}$; the coordinates are measured in meters, and $A = 4\text{ s}^{-1}$ and $B = 2\text{ s}^{-1}$. The pressure is $p_0 = 200$ kPa at point $(x, y) = (0, 0)$. Obtain an expression for the pressure field, $p(x, y)$ in terms of p_0, A, and B, and evaluate at point $(x, y) = (2, 2)$.

6.11 An incompressible liquid with a density of 1250 kg/m^3 and negligible viscosity flows steadily through a horizontal pipe of constant diameter. In a porous section of length $L = 5$ m, liquid is removed at a constant rate per unit length so that the uniform axial velocity in the pipe is $u(x) = U(1 - x/L)$, where $U = 15$ m/s. Develop expressions for and plot the pressure gradient along the centerline. Evaluate the outlet pressure if the pressure at the inlet to the porous section is 100 kPa (gage).

6.12 An incompressible liquid with a density of 900 kg/m^3 and negligible viscosity flows steadily through a horizontal pipe of constant diameter. In a porous section of length $L = 2$ m, liquid is removed at a variable rate along the length so that the uniform axial velocity in the pipe is $u(x) = Ue^{-x/L}$, where $U = 20$ m/s. Develop expressions for and plot the acceleration of a fluid particle along the centerline of the porous section and the pressure gradient along the centerline. Evaluate the outlet pressure if the pressure at the inlet to the porous section is 50 kPa (gage).

6.13 For the flow of Problem 4.123 show that the uniform radial velocity is $V_r = Q/2\pi rh$. Obtain expressions for the r component of acceleration of a fluid particle in the gap and for the pressure variation as a function of radial distance from the central holes.

6.14 The velocity field for a plane vortex sink is given by $\vec{V} = (-q/2\pi r)\hat{e}_r + (K/2\pi r)\hat{e}_\theta$, where $q = 2$ m^3/s/m and $K = 1$ m^3/s/m. The fluid density is 1000 kg/m^3. Find the acceleration at (1, 0), (1, π/2), and (2, 0). Evaluate ∇p under the same conditions.

6.15 An incompressible, inviscid fluid flows into a horizontal round tube through its porous wall. The tube is closed at the left end and the flow discharges from the tube to the atmosphere at the right end. For simplicity, consider the x component of velocity in the tube uniform across any cross section. The density of the fluid is ρ, the tube diameter and length are D and L, respectively, and the uniform inflow velocity is v_0. The flow is steady. Obtain an algebraic expression for the x component of acceleration of a fluid particle located at position x, in terms of v_0, x, and D. Find an expression for the pressure gradient, $\partial p/\partial x$, at position x. Integrate to obtain an expression for the gage pressure at $x = 0$.

6.16 An incompressible liquid with negligible viscosity and density $\rho = 1.75$ slug/ft^3 flows steadily through a horizontal pipe. The pipe cross-section area linearly varies from 15 in^2 to 2.5 in^2 over a length of 10 feet. Develop an expression for and plot the pressure gradient and pressure versus position along the pipe, if the inlet centerline velocity is 5 ft/s and inlet pressure is 35 psi. What is the exit pressure? *Hint:* Use relation

$$u\frac{\partial u}{\partial x} = \frac{1}{2}\frac{\partial}{\partial x}(u^2)$$

6.17 An incompressible liquid with negligible viscosity and density $\rho = 1250$ kg/m^3 flows steadily through a 5-m-long convergent-divergent section of pipe for which the area varies as

$$A(x) = A_0(1 + e^{-x/a} - e^{-x/2a})$$

where $A_0 = 0.25$ m^2 and $a = 1.5$ m. Plot the area for the first 5 m. Develop an expression for and plot the pressure gradient and pressure versus position along the pipe, for the first 5 m, if the inlet centerline velocity is 10 m/s and inlet pressure is 300 kPa. *Hint:* Use relation

$$u\frac{\partial u}{\partial x} = \frac{1}{2}\frac{\partial}{\partial x}(u^2)$$

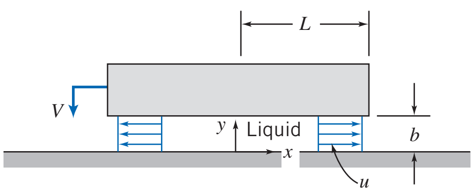
6.18 A nozzle for an incompressible, inviscid fluid of density $\rho = 1000\ \text{kg/m}^3$ consists of a converging section of pipe. At the inlet the diameter is $D_i = 100$ mm, and at the outlet the diameter is $D_o = 20$ mm. The nozzle length is $L = 500$ mm, and the diameter decreases linearly with distance x along the nozzle. Derive and plot the acceleration of a fluid particle, assuming uniform flow at each section, if the speed at the inlet is $V_i = 1$ m/s. Plot the pressure gradient through the nozzle, and find its maximum absolute value. If the pressure gradient must be no greater than 5 MPa/m in absolute value, how long would the nozzle have to be?

6.19 A diffuser for an incompressible, inviscid fluid of density $\rho = 1000\ \text{kg/m}^3$ consists of a diverging section of pipe. At the inlet the diameter is $D_i = 0.25$ m, and at the outlet the diameter is $D_o = 0.75$ m. The diffuser length is $L = 1$ m, and the diameter increases linearly with distance x along the diffuser. Derive and plot the acceleration of a fluid particle, assuming uniform flow at each section, if the speed at the inlet is $V_i = 5$ m/s. Plot the pressure gradient through the diffuser, and find its maximum value. If the pressure gradient must be no greater than 25 kPa/m, how long would the diffuser have to be?

6.20 Consider the flow of Problem 5.48. Evaluate the magnitude and direction of the net pressure force that acts on the upper plate between r_i and R, if $r_i = R/2$.

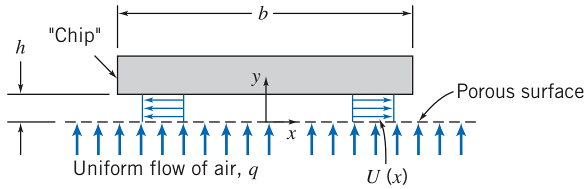
6.21 Consider again the flow field of Problem 5.65. Assume the flow is incompressible with $\rho = 1.23\ \text{kg/m}^3$ and friction is negligible. Further assume the vertical air flow velocity is $v_0 = 15$ mm/s, the half-width of the cavity is $L = 22$ mm, and its height is $h = 1.2$ mm. Calculate the pressure gradient at $(x, y) = (L, h)$. Obtain an equation for the flow streamlines in the cavity.

6.22 A liquid layer separates two plane surfaces as shown. The lower surface is stationary; the upper surface moves downward at constant speed V. The moving surface has width w, perpendicular to the plane of the diagram, and $w \gg L$. The incompressible liquid layer, of density ρ, is squeezed from between the surfaces. Assume the flow is uniform at any cross section and neglect viscosity as a first approximation. Use a suitably chosen control volume to show that $u = Vx/b$ within the gap, where $b = b_0 - Vt$. Obtain an algebraic expression for the acceleration of a fluid particle located at x. Determine the pressure gradient, $\partial p/\partial x$, in the liquid layer. Find the pressure distribution, $p(x)$. Obtain an expression for the net pressure force that acts on the upper (moving) flat surface.



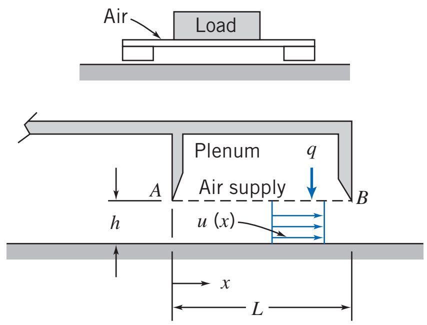
P6.22

6.23 A rectangular computer chip floats on a thin layer of air, $h = 0.5$ mm thick, above a porous surface. The chip width is $b = 40$ mm, as shown. Its length, L, is very long in the direction perpendicular to the diagram. There is no flow in the z direction. Assume flow in the x direction in the gap under the chip is uniform. Flow is incompressible, and frictional effects may be neglected. Use a suitably chosen control volume to show that $U(x) = qx/h$ in the gap. Find a general expression for the (2D) acceleration of a fluid particle in the gap in terms of $q, h, x,$ and y. Obtain an expression for the pressure gradient $\partial p/\partial x$. Assuming atmospheric pressure on the chip upper surface, find an expression for the net pressure force on the chip; is it directed upward or downward? Explain. Find the required flow rate q ($\text{m}^3/\text{s}/\text{m}^2$) and the maximum velocity, if the mass per unit length of the chip is 0.005 kg/m. Plot the pressure distribution as part of your explanation of the direction of the net force.



P6.23

6.24 Heavy weights can be moved with relative ease on air cushions by using a load pallet as shown. Air is supplied from the plenum through porous surface AB. It enters the gap vertically at uniform speed, q. Once in the gap, all air flows in the positive x direction (there is no flow across the plane at $x = 0$). Assume air flow in the gap is incompressible and uniform at each cross section, with speed $u(x)$, as shown in the enlarged view. Although the gap is narrow ($h \ll L$), neglect frictional effects as a first approximation. Use a suitably chosen control volume to show that $u(x) = qx/h$ in the gap. Calculate the acceleration of a fluid particle in the gap. Evaluate the pressure gradient, $\partial p/\partial x$, and sketch the pressure distribution within the gap. Be sure to indicate the pressure at $x = L$.



P6.24

6.25 A velocity field is given by $\vec{V} = [Ax^3 + Bxy^2]\hat{i} + [Ay^3 + Bx^2y]\hat{j}$; $A = 0.2\ \text{m}^{-2}\cdot\text{s}^{-1}$, B is a constant, and the coordinates are measured in meters. Determine the value and units for B if this velocity field is to represent an incompressible flow. Calculate the acceleration of a fluid particle at point $(x, y) = (2, 1)$. Evaluate the component of particle acceleration normal to the velocity vector at this point.

6.26 The y component of velocity in a two-dimensional incompressible flow field is given by $v = -Axy$, where v is in m/s, the coordinates are measured in meters, and $A = 1\ \text{m}^{-1}\cdot\text{s}^{-1}$. There is no velocity component or variation in the z direction. Calculate the acceleration of a fluid particle at point $(x, y) = (1, 2)$. Estimate the radius of curvature of the

streamline passing through this point. Plot the streamline and show both the velocity vector and the acceleration vector on the plot. (Assume the simplest form of the x component of velocity.)

6.27 Consider the velocity field $\vec{V} = A[x^4 - 6x^2y^2 + y^4]\hat{i} + B[x^3y - xy^3]\hat{j}$; $A = 2\ \text{m}^{-3}\cdot\text{s}^{-1}$, B is a constant, and the coordinates are measured in meters. Find B for this to be an incompressible flow. Obtain the equation of the streamline through point $(x, y) = (1, 2)$. Derive an algebraic expression for the acceleration of a fluid particle. Estimate the radius of curvature of the streamline at $(x, y) = (1, 2)$.

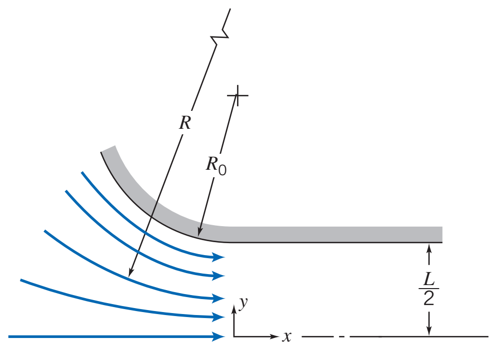
6.28 The velocity field for a plane doublet is given in Table 6.2. Find an expression for the pressure gradient at any point (r, θ).

6.29 Air flow over a stationary circular cylinder of radius a is modeled as a steady, frictionless, and incompressible flow from right to left, given by the velocity field

$$\vec{V} = U\left[\left(\frac{a}{r}\right)^2 - 1\right]\cos\theta\,\hat{e}_r + U\left[\left(\frac{a}{r}\right)^2 + 1\right]\sin\theta\,\hat{e}_\theta$$

Consider flow along the streamline forming the cylinder surface, $r = a$. Express the components of the pressure gradient in terms of angle θ. Obtain an expression for the variation of pressure (gage) on the surface of the cylinder. For $U = 75$ m/s and $a = 150$ mm, plot the pressure distribution (gage) and explain, and find the minimum pressure. Plot the speed V as a function of r along the radial line $\theta = \pi/2$ for $r > a$ (that is, directly above the cylinder), and explain.

6.30 To model the velocity distribution in the curved inlet section of a water channel, the radius of curvature of the streamlines is expressed as $R = LR_0/2y$. As an approximation, assume the water speed along each streamline is $V = 10$ m/s. Find an expression for and plot the pressure distribution from $y = 0$ to the tunnel wall at $y = L/2$, if the centerline pressure (gage) is 50 kPa, $L = 75$ mm, and $R_0 = 0.2$ m. Find the value of V for which the wall static pressure becomes 35 kPa.



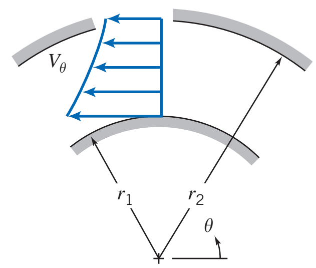
P6.30

6.31 Air at 20 psia and 100°F flows around a smooth corner at the inlet to a diffuser. The air speed is 150 ft/s, and the radius of curvature of the streamlines is 3 in. Determine the magnitude of the centripetal acceleration experienced by a fluid particle rounding the corner. Express your answer in gs. Evaluate the pressure gradient, $\partial p/\partial r$.

6.32 Repeat Example 6.1, but with the somewhat more realistic assumption that the flow is similar to a free vortex (irrotational) profile, $V_\theta = c/r$ (where c is a constant), as shown in Fig. P6.32. In doing so, prove that the flow rate is given by $Q = k\sqrt{\Delta p}$, where k is

$$k = w\ln\left(\frac{r_2}{r_1}\right)\sqrt{\frac{2r_2^2r_1^2}{\rho(r_2^2 - r_1^2)}}$$

and w is the depth of the bend.



P6.32

6.33 The velocity field in a two-dimensional, steady, inviscid flow field in the horizontal xy plane is given by $\vec{V} = (Ax + B)\hat{i} - Ay\hat{j}$, where $A = 1\ \text{s}^{-1}$ and $B = 2$ m/s; x and y are measured in meters. Show that streamlines for this flow are given by $(x + B/A)y = \text{constant}$. Plot streamlines passing through points $(x, y) = (1, 1)$, $(1, 2)$, and $(2, 2)$. At point $(x, y) = (1, 2)$, evaluate and plot the acceleration vector and the velocity vector. Find the component of acceleration along the streamline at the same point; express it as a vector. Evaluate the pressure gradient along the streamline at the same point if the fluid is air. What statement, if any, can you make about the relative value of the pressure at points (1, 1) and (2, 2)?

6.34 Using the analyses of Example 6.1 and Problem 6.32, plot the discrepancy (percent) between the flow rates obtained from assuming uniform flow and the free vortex (irrotational) profile as a function of inner radius r_1.

6.35 The x component of velocity in a two-dimensional, incompressible flow field is given by $u = Ax^2$; the coordinates are measured in feet and $A = 1\ \text{ft}^{-1}\cdot\text{s}^{-1}$. There is no velocity component or variation in the z direction. Calculate the acceleration of a fluid particle at point $(x, y) = (1, 2)$. Estimate the radius of curvature of the streamline passing through this point. Plot the streamline and show both the velocity vector and the acceleration vector on the plot. (Assume the simplest form of the y component of velocity.)

6.36 The x component of velocity in a two-dimensional incompressible flow field is given by $u = -\Lambda(x^2 - y^2)/(x^2 + y^2)^2$, where u is in m/s, the coordinates are measured in meters, and $\Lambda = 2\ \text{m}^3\cdot\text{s}^{-1}$. Show that the simplest form of the y component of velocity is given by $v = -2\Lambda xy/(x^2 + y^2)^2$. There is no velocity component or variation in the z direction. Calculate the acceleration of fluid particles at points $(x, y) = (0, 1)$, $(0, 2)$, and $(0, 3)$. Estimate the radius of curvature of the streamlines passing through these points. What does the relation among the three points and their radii of curvature suggest to you about the flow field? Verify this by

plotting these streamlines. [*Hint*: You will need to use an integrating factor.]

6.37 The x component of velocity in a two-dimensional, incompressible flow field is given by $u = Axy$; the coordinates are measured in meters and $A = 2\ \text{m}^{-1} \cdot \text{s}^{-1}$. There is no velocity component or variation in the z direction. Calculate the acceleration of a fluid particle at point $(x, y) = (2, 1)$. Estimate the radius of curvature of the streamline passing through this point. Plot the streamline and show both the velocity vector and the acceleration vector on the plot. (Assume the simplest form of the y component of velocity.)

The Bernoulli Equation

6.38 Water flows at a speed of 25 ft/s. Calculate the dynamic pressure of this flow. Express your answer in inches of mercury.

6.39 Calculate the dynamic pressure that corresponds to a speed of 100 km/hr in standard air. Express your answer in millimeters of water.

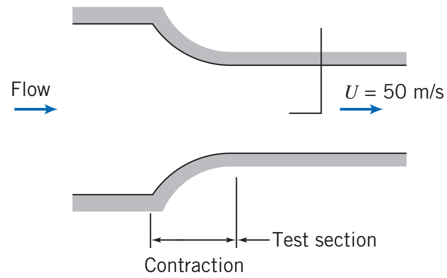
6.40 Plot the speed of air versus the dynamic pressure (in millimeters of mercury), up to a dynamic pressure of 250 mm Hg.

6.41 You present your open hand out of the window of an automobile perpendicular to the airflow. Assuming for simplicity that the air pressure on the entire front surface is stagnation pressure (with respect to automobile coordinates), with atmospheric pressure on the rear surface, estimate the net force on your hand when driving at (a) 30 mph and (b) 60 mph. Do these results roughly correspond with your experience? Do the simplifications tend to make the calculated force an over- or underestimate?

6.42 A jet of air from a nozzle is blown at right angles against a wall in which two pressure taps are located. A manometer connected to the tap directly in front of the jet shows a head of 25 mm of mercury above atmospheric. Determine the approximate speed of the air leaving the nozzle if it is at $-10°\text{C}$ and 200 kPa. At the second tap a manometer indicates a head of 5 mm of mercury above atmospheric; what is the approximate speed of the air there?

6.43 A pitot-static tube is used to measure the speed of air at standard conditions at a point in a flow. To ensure that the flow may be assumed incompressible for calculations of engineering accuracy, the speed is to be maintained at 100 m/s or less. Determine the manometer deflection, in millimeters of water, that corresponds to the maximum desirable speed.

6.44 The inlet contraction and test section of a laboratory wind tunnel are shown. The air speed in the test section is $U = 50$ m/s. A total-head tube pointed upstream indicates that the stagnation pressure on the test section centerline is 10 mm of water below atmospheric. The laboratory is maintained at atmospheric pressure and a temperature of $-5°\text{C}$. Evaluate the dynamic pressure on the centerline of the wind tunnel test section. Compute the static pressure at the same point. Qualitatively compare the static pressure at the tunnel wall with that at the centerline. Explain why the two may not be identical.



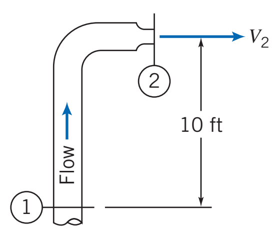
P6.44

6.45 Maintenance work on high-pressure hydraulic systems requires special precautions. A small leak can result in a high-speed jet of hydraulic fluid that can penetrate the skin and cause serious injury (therefore troubleshooters are cautioned to use a piece of paper or cardboard, *not a finger*, to search for leaks). Calculate and plot the jet speed of a leak versus system pressure, for pressures up to 40 MPa (gage). Explain how a high-speed jet of hydraulic fluid can cause injury.

6.46 An open-circuit wind tunnel draws in air from the atmosphere through a well-contoured nozzle. In the test section, where the flow is straight and nearly uniform, a static pressure tap is drilled into the tunnel wall. A manometer connected to the tap shows that static pressure within the tunnel is 45 mm of water below atmospheric. Assume that the air is incompressible, and at 25°C, 100 kPa (abs). Calculate the air speed in the wind-tunnel test section.

6.47 The wheeled cart shown in Problem 4.128 rolls with negligible resistance. The cart is to accelerate to the right. The jet speed is $V = 40$ m/s. The jet area remains constant at $A = 25\ \text{mm}^2$. Neglect viscous forces between the water and vane. When the cart attains speed $U = 15$ m/s, calculate the stagnation pressure of the water leaving the nozzle with respect to a fixed observer, the stagnation pressure of the water jet leaving the nozzle with respect to an observer on the vane, the absolute velocity of the jet leaving the vane with respect to a fixed observer, and the stagnation pressure of the jet leaving the vane with respect to a fixed observer. How would viscous forces affect the latter stagnation pressure, i.e., would viscous forces increase, decrease, or leave unchanged this stagnation pressure? Justify your answer.

6.48 Water flows steadily up the vertical 1-in.-diameter pipe and out the nozzle, which is 0.5 in. in diameter, discharging to atmospheric pressure. The stream velocity at the nozzle exit must be 30 ft/s. Calculate the minimum gage pressure required at section 1. If the device were inverted, what would be the required minimum pressure at section 1 to maintain the nozzle exit velocity at 30 ft/s?



P6.48

6.49 Water flows in a circular duct. At one section the diameter is 0.3 m, the static pressure is 260 kPa (gage), the velocity is

3 m/s, and the elevation is 10 m above ground level. At a section downstream at ground level, the duct diameter is 0.15 m. Find the gage pressure at the downstream section if frictional effects may be neglected.

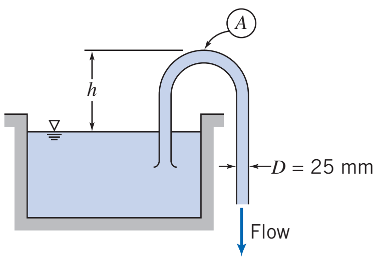
6.50 You are on a date. Your date runs out of gas unexpectedly. You come to your own rescue by siphoning gas from another car. The height difference for the siphon is about 1 ft. The hose diameter is 0.5 in. What is your gasoline flow rate?

6.51 You (a young person of legal drinking age) are making homemade beer. As part of the process you have to siphon the wort (the fermenting beer with sediment at the bottom) into a clean tank using a 5-mm ID tubing. Being a young engineer, you're curious about the flow you can produce. Find an expression for and plot the flow rate Q (liters per minute) versus the differential in height h (millimeters) between the wort free surface and the location of the hose exit. Find the value of h for which $Q = 2$ L/min.

6.52 A tank at a pressure of 50 kPa (gage) gets a pinhole rupture and benzene shoots into the air. Ignoring losses, to what height will the benzene rise?

6.53 A can of Coke (you're not sure if it's diet or regular) has a small pinhole leak in it. The Coke sprays vertically into the air to a height of 0.5 m. What is the pressure inside the can of Coke? (Estimate for both kinds of Coke.)

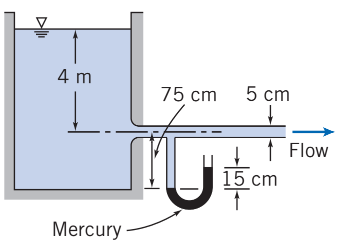
6.54 The water flow rate through the siphon is 5 L/s, its temperature is 20°C, and the pipe diameter is 25 mm. Compute the maximum allowable height, h, so that the pressure at point A is above the vapor pressure of the water. (Assume the flow is frictionless.)



P6.54

6.55 A stream of liquid moving at low speed leaves a nozzle pointed directly downward. The velocity may be considered uniform across the nozzle exit and the effects of friction may be ignored. At the nozzle exit, located at elevation z_0, the jet velocity and area are V_0 and A_0, respectively. Determine the variation of jet area with elevation.

6.56 Water flows from a very large tank through a 5-cm-diameter tube. The dark liquid in the manometer is mercury. Estimate the velocity in the pipe and the rate of discharge from the tank. (Assume the flow is frictionless.)



P6.56

6.57 In a laboratory experiment, water flows radially outward at moderate speed through the space between circular plane parallel disks. The perimeter of the disks is open to the atmosphere. The disks have diameter $D = 150$ mm and the spacing between the disks is $h = 0.8$ mm. The measured mass flow rate of water is $\dot{m} = 305$ g/s. Assuming frictionless flow in the space between the disks, estimate the theoretical static pressure between the disks at radius $r = 50$ mm. In the laboratory situation, where *some* friction is present, would the pressure measured at the same location be above or below the theoretical value? Why?

6.58 Consider frictionless, incompressible flow of air over the wing of an airplane flying at 200 km/hr. The air approaching the wing is at 65 kPa and −10°C. At a certain point in the flow, the pressure is 60 kPa. Calculate the speed of the air relative to the wing at this point and the absolute air speed.

6.59 A speedboat on hydrofoils is moving at 20 m/s in a freshwater lake. Each hydrofoil is 3 m below the surface. Assuming, as an approximation, frictionless, incompressible flow, find the stagnation pressure (gage) at the front of each hydrofoil. At one point on a hydrofoil, the pressure is −75 kPa (gage). Calculate the speed of the water relative to the hydrofoil at this point and the absolute water speed.

6.60 A mercury barometer is carried in a car on a day when there is no wind. The temperature is 20°C and the corrected barometer height is 761 mm of mercury. One window is open slightly as the car travels at 105 km/hr. The barometer reading in the moving car is 5 mm lower than when the car is stationary. Explain what is happening. Calculate the local speed of the air flowing past the window, *relative to* the automobile.

6.61 A fire nozzle is coupled to the end of a hose with inside diameter $D = 3$ in. The nozzle is contoured smoothly and has outlet diameter $d = 1$ in. The design inlet pressure for the nozzle is $p_1 = 100$ psi (gage). Evaluate the maximum flow rate the nozzle could deliver.

6.62 A racing car travels at 235 mph along a straightaway. The team engineer wishes to locate an air inlet on the body of the car to obtain cooling air for the driver's suit. The plan is to place the inlet at a location where the air speed is 60 mph along the surface of the car. Calculate the static pressure at the proposed inlet location. Express the pressure rise above ambient as a fraction of the freestream dynamic pressure.

6.63 Steady, frictionless, and incompressible flow from left to right over a stationary circular cylinder, of radius a, is represented by the velocity field

$$\vec{V} = U\left[1 - \left(\frac{a}{r}\right)^2\right]\cos\theta\,\hat{e}_r - U\left[1 + \left(\frac{a}{r}\right)^2\right]\sin\theta\,\hat{e}_\theta$$

Obtain an expression for the pressure distribution along the streamline forming the cylinder surface, $r = a$. Determine the locations where the static pressure on the cylinder is equal to the freestream static pressure.

6.64 The velocity field for a plane source at a distance h above an infinite wall aligned along the x axis was given in Problem 6.8. Using the data from that problem, plot the

pressure distribution along the wall from $x = -10h$ to $x = +10h$ (assume the pressure at infinity is atmospheric). Find the net force on the wall if the pressure on the lower surface is atmospheric. Does the force tend to pull the wall towards the source, or push it away?

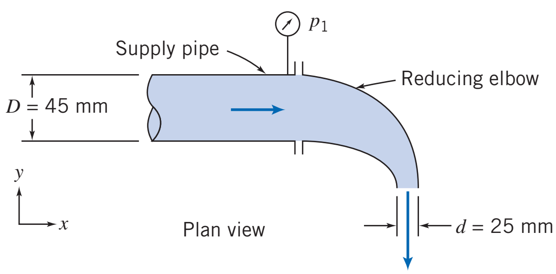
6.65 The velocity field for a plane doublet is given in Table 6.2. If $\Lambda = 3\ \text{m}^3 \cdot \text{s}^{-1}$, the fluid density is $\rho = 1.5\ \text{kg/m}^3$, and the pressure at infinity is 100 kPa, plot the pressure along the x axis from $x = -2.0$ m to -0.5 m and $x = 0.5$ m to 2.0 m.

6.66 A smoothly contoured nozzle, with outlet diameter $d = 20$ mm, is coupled to a straight pipe by means of flanges. Water flows in the pipe, of diameter $D = 50$ mm, and the nozzle discharges to the atmosphere. For steady flow and neglecting the effects of viscosity, find the volume flow rate in the pipe corresponding to a calculated axial force of 45.5 N needed to keep the nozzle attached to the pipe.

6.67 A fire nozzle is coupled to the end of a hose with inside diameter $D = 75$ mm. The nozzle is smoothly contoured and its outlet diameter is $d = 25$ mm. The nozzle is designed to operate at an inlet water pressure of 700 kPa (gage). Determine the design flow rate of the nozzle. (Express your answer in L/s.) Evaluate the axial force required to hold the nozzle in place. Indicate whether the hose coupling is in tension or compression.

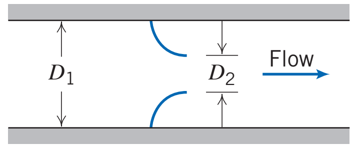
6.68 Water flows steadily through a 3.25-in.-diameter pipe and discharges through a 1.25-in.-diameter nozzle to atmospheric pressure. The flow rate is 24.5 gpm. Calculate the minimum static pressure required in the pipe to produce this flow rate. Evaluate the axial force of the nozzle assembly on the pipe flange.

6.69 Water flows steadily through the reducing elbow shown. The elbow is smooth and short, and the flow accelerates; so the effect of friction is small. The volume flow rate is $Q = 2.5$ L/s. The elbow is in a horizontal plane. Estimate the gage pressure at section 1. Calculate the x component of the force exerted by the reducing elbow on the supply pipe.



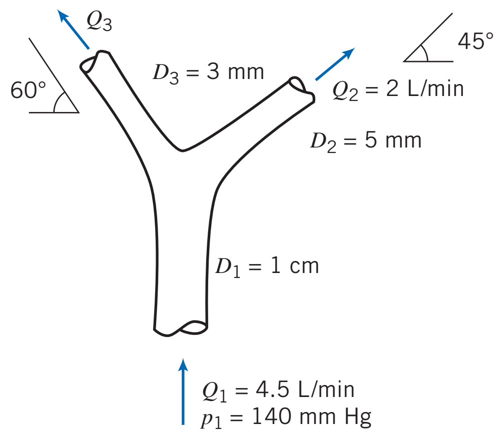
P6.69

6.70 A flow nozzle is a device for measuring the flow rate in a pipe. This particular nozzle is to be used to measure low-speed air flow for which compressibility may be neglected. During operation, the pressures p_1 and p_2 are recorded, as well as upstream temperature, T_1. Find the mass flow rate in terms of $\Delta p = p_2 - p_1$ and T_1, the gas constant for air, and device diameters D_1 and D_2. Assume the flow is frictionless. Will the actual flow be more or less than this predicted flow? Why?



P6.70

6.71 The branching of a blood vessel is shown. Blood at a pressure of 140 mm Hg flows in the main vessel at 4.5 L/min. Estimate the blood pressure in each branch, assuming that blood vessels behave as rigid tubes, that we have frictionless flow, and that the vessel lies in the horizontal plane. What is the force generated at the branch by the blood? You may approximate blood to have a density of 1060 kg/m^3.



P6.71

6.72 An object, with a flat horizontal lower surface, moves downward into the jet of the spray system of Problem 4.81 with speed $U = 5$ ft/s. Determine the minimum supply pressure needed to produce the jet leaving the spray system at $V = 15$ ft/s. Calculate the maximum pressure exerted by the liquid jet on the flat object at the instant when the object is $h = 1.5$ ft above the jet exit. Estimate the force of the water jet on the flat object.

6.73 A water jet is directed upward from a well-designed nozzle of area $A_1 = 600\ \text{mm}^2$; the exit jet speed is $V_1 = 6.3$ m/s. The flow is steady and the liquid stream does not break up. Point 2 is located $H = 1.55$ m above the nozzle exit plane. Determine the velocity in the undisturbed jet at point 2. Calculate the pressure that would be sensed by a stagnation tube located there. Evaluate the force that would be exerted on a flat plate placed normal to the stream at point 2. Sketch the pressure distribution on the plate.

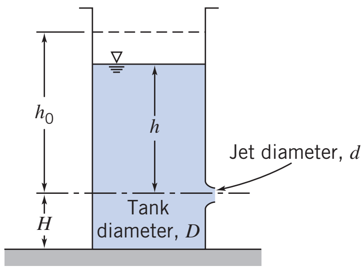
6.74 Water flows out of a kitchen faucet of 1.25 cm diameter at the rate of 0.1 L/s. The bottom of the sink is 45 cm below the faucet outlet. Will the cross-sectional area of the fluid stream increase, decrease, or remain constant between the faucet outlet and the bottom of the sink? Explain briefly. Obtain an expression for the stream cross section as a function of distance y above the sink bottom. If a plate is held directly under the faucet, how will the force required to hold the plate in a horizontal position vary with height above the sink? Explain briefly.

6.75 An old magic trick uses an empty thread spool and a playing card. The playing card is placed against the bottom of

the spool. Contrary to intuition, when one blows downward through the central hole in the spool, the card is not blown away. Instead it is "sucked" up against the spool. Explain.

6.76 A horizontal axisymmetric jet of air with 0.4 in. diameter strikes a stationary vertical disk of 7.5 in. diameter. The jet speed is 225 ft/s at the nozzle exit. A manometer is connected to the center of the disk. Calculate (a) the deflection, if the manometer liquid has SG = 1.75, (b) the force exerted by the jet on the disk, and (c) the force exerted on the disk if it is assumed that the stagnation pressure acts on the entire forward surface of the disk. Sketch the streamline pattern and plot the distribution of pressure on the face of the disk.

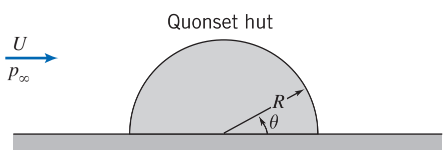
6.77 The tank, of diameter D, has a well-rounded nozzle with diameter d. At $t = 0$, the water level is at height h_0. Develop an expression for dimensionless water height, h/h_0, at any later time. For $D/d = 10$, plot h/h_0 as a function of time with h_0 as a parameter for $0.1 \leq h_0 \leq 1$ m. For $h_0 = 1$ m, plot h/h_0 as a function of time with D/d as a parameter for $2 \leq D/d \leq 10$.



P6.77

6.78 The water level in a large tank is maintained at height H above the surrounding level terrain. A rounded nozzle placed in the side of the tank discharges a horizontal jet. Neglecting friction, determine the height h at which the orifice should be placed so the water strikes the ground at the maximum horizontal distance X from the tank. Plot jet speed V and distance X as functions of h ($0 < h < H$).

6.79 The flow over a Quonset hut may be approximated by the velocity distribution of Problem 6.63 with $0 \leq \theta \leq \pi$. During a storm the wind speed reaches 100 km/hr; the outside temperature is 5°C. A barometer inside the hut reads 720 mm of mercury; pressure p_∞ is also 720 mm Hg. The hut has a diameter of 6 m and a length of 18 m. Determine the net force tending to lift the hut off its foundation.



P6.79

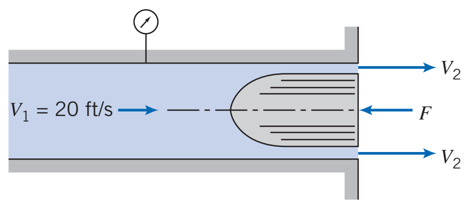
6.80 Many recreation facilities use inflatable "bubble" structures. A tennis bubble to enclose four courts is shaped roughly as a circular semicylinder with a diameter of 50 ft and a length of 50 ft. The blowers used to inflate the structure can maintain the air pressure inside the bubble at 0.75 in. of water above ambient pressure. The bubble is subjected to a wind that blows at 35 mph in a direction perpendicular to the axis of the semicylindrical shape. Using polar coordinates, with angle θ measured from the ground on the upwind side of the structure, the resulting pressure distribution may be expressed as

$$\frac{p - p_\infty}{\frac{1}{2}\rho V_\infty^2} = 1 - 4\sin^2\theta$$

where p is the pressure at the surface, p_∞ the atmospheric pressure, and V_w the wind speed. Determine the net vertical force exerted on the structure.

6.81 High-pressure air forces a stream of water from a tiny, rounded orifice, of area A, in a tank. The pressure is high enough that gravity may be neglected. The air expands slowly, so that the expansion may be considered isothermal. The initial volume of air in the tank is $V\!\!\!-_0$. At later instants the volume of air is $V\!\!\!-(t)$; the total volume of the tank is $V\!\!\!-_t$. Obtain an algebraic expression for the mass flow rate of water leaving the tank. Find an algebraic expression for the rate of change in mass of the water inside the tank. Develop an ordinary differential equation and solve for the water mass in the tank at any instant. If $V\!\!\!-_0 = 5$ m^3, $V\!\!\!-_t = 10$ m^3, $A = 25$ mm^2, and $p_0 = 1$ MPa, plot the water mass in the tank versus time for the first forty minutes.

6.82 Water flows at low speed through a circular tube with inside diameter of 2 in. A smoothly contoured body of 1.5 in. diameter is held in the end of the tube where the water discharges to atmosphere. Neglect frictional effects and assume uniform velocity profiles at each section. Determine the pressure measured by the gage and the force required to hold the body.



P6.82

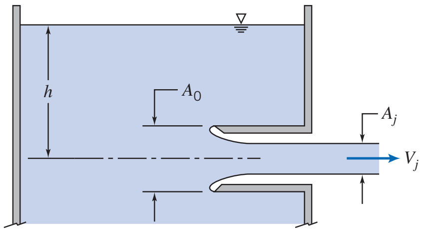
6.83 Repeat Problem 6.81 assuming the air expands so rapidly that the expansion may be treated as adiabatic.

6.84 Describe the pressure distribution on the exterior of a multistory building in a steady wind. Identify the locations of the maximum and minimum pressures on the outside of the building. Discuss the effect of these pressures on infiltration of outside air into the building.

6.85 Imagine a garden hose with a stream of water flowing out through a nozzle. Explain why the end of the hose may be unstable when held a half meter or so from the nozzle end.

6.86 An aspirator provides suction by using a stream of water flowing through a venturi. Analyze the shape and dimensions of such a device. Comment on any limitations on its use.

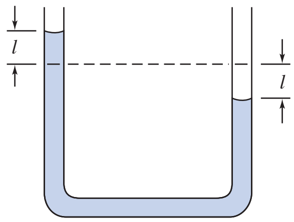
6.87 A tank with a *reentrant* orifice called a *Borda mouthpiece* is shown. The fluid is inviscid and incompressible. The reentrant orifice essentially eliminates flow along the tank walls, so the pressure there is nearly hydrostatic. Calculate the *contraction coefficient*, $C_c = A_j/A_0$. *Hint:* Equate the unbalanced hydrostatic pressure force and momentum flux from the jet.



P6.87

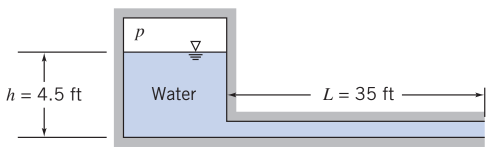
Unsteady Bernoulli Equation

***6.88** Apply the unsteady Bernoulli equation to the U-tube manometer of constant diameter shown. Assume that the manometer is initially deflected and then released. Obtain a differential equation for l as a function of time.



P6.88

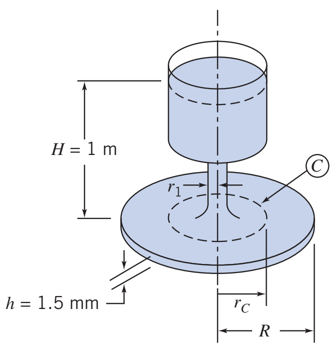
***6.89** Compressed air is used to accelerate water from a tube. Neglect the velocity in the reservoir and assume the flow in the tube is uniform at any section. At a particular instant, it is known that $V = 6$ ft/s and $dV/dt = 7.5$ ft/s^2. The cross-sectional area of the tube is $A = 32$ in.^{2}. Determine the pressure in the tank at this instant.



P6.89, P6.90, P6.92

***6.90** If the water in the pipe in Problem 6.89 is initially at rest and the air pressure is 3 psig, what will be the initial acceleration of the water in the pipe?

***6.91** Consider the reservoir and disk flow system with the reservoir level maintained constant. Flow between the disks is started from rest at $t = 0$. Evaluate the rate of change of volume flow rate at $t = 0$, if $r_1 = 50$ mm.



P6.91

***6.92** If the water in the pipe of Problem 6.89 is initially at rest, and the air pressure is maintained at 1.5 psig, derive a differential equation for the velocity V in the pipe as a function of time, integrate, and plot V versus t for $t = 0$ to 5 s.

***6.93** Consider the tank of Problem 4.46. Using the Bernoulli equation for unsteady flow along a streamline, evaluate the minimum diameter ratio, D/d, required to justify the assumption that flow from the tank is quasi-steady.

***6.94** Two circular disks, of radius R, are separated by distance b. The upper disk moves toward the lower one at constant speed V. The space between the disks is filled with a frictionless, incompressible fluid, which is squeezed out as the disks come together. Assume that, at any radial section, the velocity is uniform across the gap width b. However, note that b is a function of time. The pressure surrounding the disks is atmospheric. Determine the gage pressure at $r = 0$.

Energy Grade Line And Hydraulic Grade Line

6.95 Carefully sketch the energy grade lines (EGL) and hydraulic grade lines (HGL) for the system shown in Fig. 6.6 if the pipe is horizontal (i.e., the outlet is at the base of the reservoir), and a water turbine (extracting energy) is located at point 2, or at point 3. In Chapter 8 we will investigate the effects of friction on internal flows. Can you anticipate and sketch the effect of friction on the EGL and HGL for the two cases?

6.96 Carefully sketch the energy grade lines (EGL) and hydraulic grade lines (HGL) for the system shown in Fig. 6.6 if a pump (adding energy to the fluid) is located at point 2, or at point 3, such that flow is *into* the reservoir. In Chapter 8 we will investigate the effects of friction on internal flows. Can you anticipate and sketch the effect of friction on the EGL and HGL for the two cases?

Irrotational Flow

***6.97** Determine whether the Bernoulli equation can be applied between different radii for the vortex flow fields (a) $\vec{V} = \omega r\,\hat{e}_\theta$ and (b) $\vec{V} = \hat{e}_\theta\, K/2\pi r$.

***6.98** Consider a two-dimensional fluid flow: $u = ax + by$ and $v = cx + dy$, where a, b, c and d are constant. If the flow is incompressible and irrotational, find the relationships among a, b, c, and d. Find the stream function and velocity potential function of this flow.

***6.99** Consider the flow represented by the stream function $\psi = Ax^2y$, where A is a dimensional constant equal to 2.5 $\text{m}^{-1}\cdot\text{s}^{-1}$. The density is 1200 kg/m^3. Is the flow rotational? Can the pressure difference between points $(x, y) = (1, 4)$ and $(2, 1)$ be evaluated? If so, calculate it, and if not, explain why.

***6.100** The velocity field for a two-dimensional flow is $\vec{V} = (Ax - By)t\hat{\imath} - (Bx + Ay)t\hat{\jmath}$, where $A = 1\ \text{s}^{-2}$ $B = 2\ \text{s}^{-2}$, t is in seconds, and the coordinates are measured in meters. Is this a possible incompressible flow? Is the flow steady or unsteady? Show that the flow is irrotational and derive an expression for the velocity potential.

***6.101** Using Table 6.2, find the stream function and velocity potential for a plane source, of strength q, near a 90° corner.

*These problems require material from sections that may be omitted without loss of continuity in the text material.

The source is equidistant h from each of the two infinite planes that make up the corner. Find the velocity distribution along one of the planes, assuming $p = p_0$ at infinity. By choosing suitable values for q and h, plot the streamlines and lines of constant velocity potential. (*Hint*: Use the *Excel* workbook of Example 6.10.)

***6.102** The flow field for a plane source at a distance h above an infinite wall aligned along the x axis is given by

$$\vec{V} = \frac{q}{2\pi[x^2 + (y-h)^2]}[x\hat{i} + (y-h)\hat{j}] + \frac{q}{2\pi[x^2 + (y+h)^2]}[x\hat{i} + (y+h)\hat{j}]$$

where q is the strength of the source. The flow is irrotational and incompressible. Derive the stream function and velocity potential. By choosing suitable values for q and h, plot the streamlines and lines of constant velocity potential. (*Hint*: Use the *Excel* workbook of Example 6.10.)

***6.103** Using Table 6.2, find the stream function and velocity potential for a plane vortex, of strength K, near a 90° corner. The vortex is equidistant h from each of the two infinite planes that make up the corner. Find the velocity distribution along one of the planes, assuming $p = p_0$ at infinity. By choosing suitable values for K and h, plot the streamlines and lines of constant velocity potential. (*Hint*: Use the *Excel* workbook of Example Problem 6.10.)

***6.104** The stream function of a flow field is $\psi = Ax^2y - By^3$, where $A = 1\ \text{m}^{-1}\cdot\text{s}^{-1}$, $B = \frac{1}{3}\ \text{m}^{-1}\cdot\text{s}^{-1}$, and the coordinates are measured in meters. Find an expression for the velocity potential.

***6.105** A flow field is represented by the stream function $\psi = x^5 - 10x^3y^2 + 5xy^4$. Find the corresponding velocity field. Show that this flow field is irrotational and obtain the potential function.

***6.106** The stream function of a flow field is $\psi = Ax^3 - Bxy^2$, where $A = 1\ \text{m}^{-1}\cdot\text{s}^{-1}$ and $B = 3\ \text{m}^{-1}\cdot\text{s}^{-1}$, and coordinates are measured in meters. Find an expression for the velocity potential.

***6.107** The stream function of a flow field is $\psi = Ax^3 + B(xy^2 + x^2 - y^2)$, where ψ, x, y, A, and B are all dimensionless. Find the relation between A and B for this to be an irrotational flow. Find the velocity potential.

***6.108** A flow field is represented by the stream function $\psi = x^5 - 15x^4y^2 + 15x^2y^4 - y^6$. Find the corresponding velocity field. Show that this flow field is irrotational and obtain the potential function.

***6.109** Consider the flow field represented by the potential function $\phi = Ax^2 + Bxy - Ay^2$. Verify that this is an incompressible flow and determine the corresponding stream function.

***6.110** Consider the flow field presented by the potential function $\phi = x^5 - 10x^3y^2 + 5xy^4 - x^2 + y^2$. Verify that this is an incompressible flow, and obtain the corresponding stream function.

***6.111** Consider the flow field presented by the potential function $\phi = x^6 - 15x^4y^2 + 15x^2y^4 - y^6$. Verify that this is an incompressible flow and obtain the corresponding stream function.

***6.112** Show by expanding and collecting real and imaginary terms that $f = z^6$ (where z is the complex number $z = x + iy$) leads to a valid velocity potential (the real part of f) and a corresponding stream function (the negative of the imaginary part of f) of an irrotational and incompressible flow. Then show that the real and imaginary parts of df/dz yield $-u$ and v, respectively.

***6.113** Show that *any* differentiable function $f(z)$ of the complex number $z = x + iy$ leads to a valid potential (the real part of f) and a corresponding stream function (the negative of the imaginary part of f) of an incompressible, irrotational flow. To do so, prove using the chain rule that $f(z)$ automatically satisfies the Laplace equation. Then show that $df/dz = -u + iv$.

***6.114** Consider the flow field represented by the velocity potential $\phi = Ax + Bx^2 - By^2$, where $A = 1\ \text{m}\cdot\text{s}^{-1}$, $B = 1\ \text{m}^{-1}\cdot\text{s}^{-1}$, and the coordinates are measured in meters. Obtain expressions for the velocity field and the stream function. Calculate the pressure difference between the origin and point $(x, y) = (1, 2)$.

***6.115** A flow field is represented by the potential function $\phi = Ay^3 - Bx^2y$, where $A = 1/3\ \text{m}^{-1}\cdot\text{s}^{-1}$, $B = 1\ \text{m}^{-1}\cdot\text{s}^{-1}$, and the coordinates are measured in meters. Obtain an expression for the magnitude of the velocity vector. Find the stream function for the flow. Plot the streamlines and potential lines, and visually verify that they are orthogonal. (*Hint*: Use the *Excel* workbook of Example 6.10.)

***6.116** An incompressible flow field is characterized by the stream function $\psi = 3Ax^2y - Ay^3$, where $A = 1\ \text{m}^{-1}\cdot\text{s}^{-1}$. Show that this flow field is irrotational. Derive the velocity potential for the flow. Plot the streamlines and potential lines, and visually verify that they are orthogonal. (*Hint*: Use the *Excel* workbook of Example 6.10.)

***6.117** A certain irrotational flow field in the xy plane has the stream function $\psi = Bxy$, where $B = 0.25\ \text{s}^{-1}$, and the coordinates are measured in meters. Determine the rate of flow between points $(x, y) = (2, 2)$ and $(3, 3)$. Find the velocity potential for this flow. Plot the streamlines and potential lines, and visually verify that they are orthogonal. (*Hint*: Use the *Excel* workbook of Example 6.10.)

***6.118** The velocity distribution in a two-dimensional, steady, inviscid flow field in the xy plane is $\vec{V} = (Ax + B)\hat{i} + (C - Ay)\hat{j}$, where $A = 3\ \text{s}^{-1}$, $B = 6$ m/s, $C = 4$ m/s, and the coordinates are measured in meters. The body force distribution is $\vec{B} = -g\hat{k}$ and the density is 825 kg/m^3. Does this represent a possible incompressible flow field? Plot a few streamlines in the upper half plane. Find the stagnation point(s) of the flow field. Is the flow irrotational? If so, obtain the potential function. Evaluate the pressure difference between the origin and point $(x, y, z) = (2, 2, 2)$.

*These problems require material from sections that may be omitted without loss of continuity in the text material.

***6.119** Consider flow around a circular cylinder with freestream velocity from right to left and a counterclockwise free vortex. Show that the lift force on the cylinder can be expressed as $F_L = -\rho U \Gamma$, as illustrated in Example 6.12.

***6.120** Consider the flow past a circular cylinder, of radius a, used in Example 6.11. Show that $V_r = 0$ along the lines $(r, \theta) = (r, \pm\pi/2)$. Plot V_θ/U versus radius for $r \geq a$, along the line $(r, \theta) = (r, \pi/2)$. Find the distance beyond which the influence of the cylinder is less than 1 percent of U.

***6.121** A crude model of a tornado is formed by combining a sink, of strength $q = 2800\ \text{m}^2/\text{s}$, and a free vortex, of strength $K = 5600\ \text{m}^2/\text{s}$. Obtain the stream function and velocity potential for this flow field. Estimate the radius beyond which the flow may be treated as incompressible. Find the gage pressure at that radius.

***6.122** A source and a sink with strengths of equal magnitude, $q = 3\pi\ \text{m}^2/\text{s}$, are placed on the x axis at $x = -a$ and $x = a$, respectively. A uniform flow, with speed $U = 20$ m/s, in the positive x direction, is added to obtain the flow past a Rankine body. Obtain the stream function, velocity potential, and velocity field for the combined flow. Find the value of $\psi =$ constant on the stagnation streamline. Locate the stagnation points if $a = 0.3$ m.

***6.123** Consider again the flow past a Rankine body of Problem 6.122. The half-width, h, of the body in the y direction is given by the transcendental equation

$$\frac{h}{a} = \cot\left(\frac{\pi U h}{q}\right)$$

Evaluate the half-width, h. Find the local velocity and the pressure at points $(x, y) = (0, \pm h)$. Assume the fluid density is that of standard air.

***6.124** A flow field is formed by combining a uniform flow in the positive x direction, with $U = 10$ m/s, and a counterclockwise vortex, with strength $K = 16\pi\ \text{m}^2/\text{s}$, located at the origin. Obtain the stream function, velocity potential, and velocity field for the combined flow. Locate the stagnation point(s) for the flow. Plot the streamlines and potential lines. (*Hint*: Use the *Excel* workbook of Example 6.10.)

***6.125** Consider the flow field formed by combining a uniform flow in the positive x direction with a sink located at the origin. Let $U = 50$ m/s and $q = 90\ \text{m}^2/\text{s}$. Use a suitably chosen control volume to evaluate the net force per unit depth needed to hold in place (in standard air) the surface shape formed by the stagnation streamline.

***6.126** Consider the flow field formed by combining a uniform flow in the positive x direction and a source located at the origin. Obtain expressions for the stream function, velocity potential, and velocity field for the combined flow. If $U = 25$ m/s, determine the source strength if the stagnation point is located at $x = -1$ m. Plot the streamlines and potential lines. (*Hint*: Use the *Excel* workbook of Example 6.10.)

***6.127** Consider the flow field formed by combining a uniform flow in the positive x direction and a source located at the origin. Let $U = 30$ m/s and $q = 150\ \text{m}^2/\text{s}$. Plot the ratio of the local velocity to the freestream velocity as a function of θ along the stagnation streamline. Locate the points on the stagnation streamline where the velocity reaches its maximum value. Find the gage pressure there if the fluid density is $1.2\ \text{kg/m}^3$.

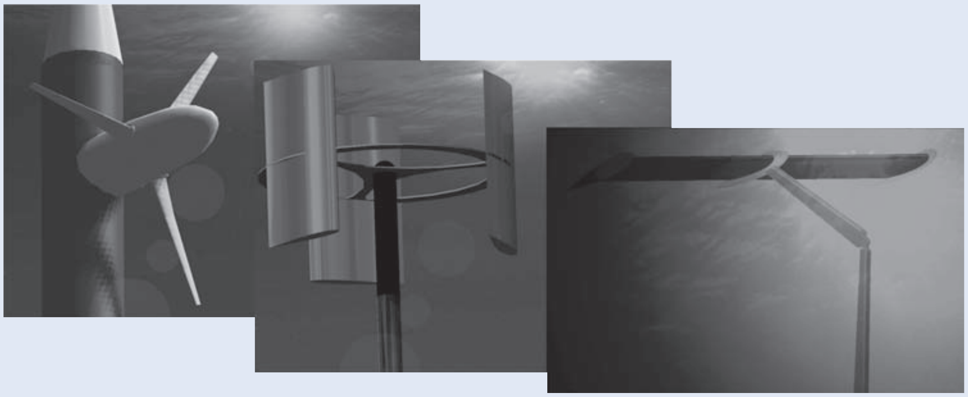
7

Dimensional Analysis and Similitude

Case Study in Energy and the Environment

Ocean Current Power: *The Vivace*

We have so far presented *Case Studies in Energy and the Environment* mostly on wave power, but many developments are taking place in ocean current power—actually, in the power available wherever there is a current, such as in estuaries and rivers, not just in the ocean. Plenty of power is available. Although ocean and river currents move slowly compared to typical wind speeds, they carry a great deal of energy because water is about 1000 times as dense as air, and the energy flux in a current is directly proportional to density. Hence water moving at 10 mph exerts about the same amount of force as a 100-mph wind. Ocean and river currents thus contain an enormous amount of energy that can be captured and converted to a usable form. For example, near the surface of the Florida Straits Current, the relatively constant extractable energy density is about 1 kW/m^2 of flow area. It has been estimated that capturing just 1/1000th of the available energy from the Gulf Stream could supply Florida with 35 percent of its electrical needs.

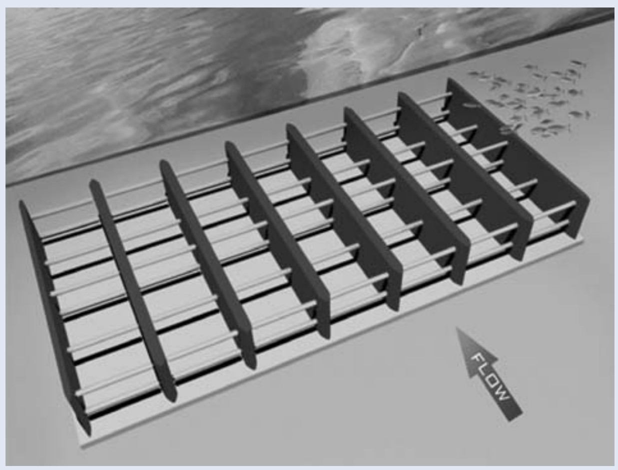


A horizontal- and a vertical-axis turbine, and an oscillating foil device (Courtesy the University of Strathclyde)

Ocean current energy is at an early stage of development, and only a small number of prototypes and demonstration units have so far been tested. A team of young engineers at the University of Strathclyde in Scotland recently did a survey of current developments. They found that perhaps the most obvious approach is to use submerged turbines. The first figure shows a horizontal-axis turbine (which is similar to a wind turbine) and a vertical-axis turbine. In each case, columns, cables, or anchors are required to keep the turbines stationary relative to the currents with which they interact. For example, they may be tethered with cables, in such a way that the current interacts with the turbine to maintain its location and stability; this is analogous to underwater kite flying in which the turbine plays the role of kite and the ocean-bottom anchor, the role of kite flyer. Turbines can include venturi-shaped shrouds around the blades to increase the flow speed and power output from the turbine. In regions with powerful currents over a large area, turbines could be assembled in clusters, similar to wind turbine farms. Space would be needed between the water turbines to eliminate wake-interaction effects and to allow access by maintenance vessels. The engineers at Strathclyde also discuss the third device shown in the figure, an oscillating foil design, in which a hydrofoil's angle of attack would be repeatedly adjusted to generate a lift force that is upward, then downward. The mechanism and controls would use this oscillating force to generate power. The advantage of this design is that there are no rotating parts that could become fouled, but the disadvantage is that the control systems involved would be quite complex.

For ocean current energy to be commercially successful, a number of technical challenges need to be addressed, including cavitation problems, prevention of marine growth buildup on turbine blades, and corrosion resistance. Environmental concerns include the protection of wildlife (fish and marine mammals) from turning turbine blades.

As the research in these types of turbines and foils continues, engineers are also looking at alternative devices. A good example is the work of Professor Michael Bernitsas, of the Department of Naval Architecture and Marine Engineering at the University of Michigan. He has developed a novel device, called the *Vivace Converter*, which uses the well-known phenomenon of vortex-induced vibrations to extract power from a flowing current. We are all familiar with vortex-induced vibrations, in which an object in a flow is made to vibrate due to vortices shedding first from one side and then the other side of the object's rear. For example, cables or wires often vibrate in the wind, sometimes sufficiently to make noise (*Aeolian tones*); many factory chimneys and car antennas have a spiral surface built into them specifically to suppress this vibration. Another famous example is the collapse of the Tacoma Narrows Bridge in Washington State in 1940, which many engineers believe was due to vortex-shedding of cross winds (a quite scary, but



The *Vivace Converter* (Courtesy Professor Michael Bernitsas)

fascinating, video of this can easily be found on the Internet). Professor Bernitas has made a source of energy from a phenomenon that is usually a nuisance or a danger!

The figure shows a conceptualization of his device, which consists of an assemblage of horizontal submerged cylinders. As the current flows across these, vortex shedding occurs, generating an oscillating up-and-down force on each cylinder. Instead of the cylinders being rigidly mounted, they are attached to a hydraulic system designed in such a way that, as the cylinders are forced up and down, they generate power. Whereas existing turbine systems need a current of about 5 knots to operate efficiently, the Vivace can generate energy using currents that are as slow as 1 knot (most of the earth's currents are slower than 3 knots). The device also does not obstruct views or access on the water's surface because it can be installed on the river or ocean floor. It's probable that this new technology is gentler on aquatic life because it is slow moving and mimics the natural vortex patterns created by the movement of swimming fish. An installation of 1×1.5 km (less than 1/2 mi^2) in a current of 3 knots could generate enough power for 100,000 homes. A prototype, funded by the U.S. Department of Energy and the Office Naval Research, is currently operating in the Marine Hydrodynamics Laboratory at the University of Michigan. The phenomenon of vortex shedding is discussed in Chapter 9; the vortex flow meter, which exploits the phenomenon to measure flow rate, is discussed in Chapter 8. We will discuss airfoil design in Chapter 9 and concepts behind the operation of turbines and propellers in Chapter 10.

In previous chapters we have mentioned several instances in which we claim a simplified flow exists. For example, we have stated that a flow with typical speed V will be essentially incompressible if the Mach number, $M \equiv V/c$ (where c is the speed of sound), is less than about 0.3 and that we can neglect viscous effects in most of a flow if the Reynolds number, $Re = \rho VL/\mu$ (L is a typical or "characteristic" size scale of the flow), is "large." We will also make extensive use of the Reynolds number based on pipe diameter, D ($Re = \rho VD/\mu$), to predict with a high degree of accuracy whether the pipe flow is laminar or turbulent. It turns out that there are many such interesting dimensionless groupings in engineering science—for example, in heat transfer, the value of the Biot number, $Bi = hL/k$, of a hot body, size L and conductivity k, indicates whether that body will tend to cool on the outside surface first or will basically cool uniformly when it's plunged into a cool fluid with convection coefficient h. (Can you figure out what a high Bi number predicts?) How do we obtain these groupings, and why do their values have such powerful predictive power?

The answers to these questions will be provided in this chapter when we introduce the method of dimensional analysis. This is a technique for gaining insight into fluid flows (in fact, into many engineering and scientific phenomena) before we do either extensive theoretical analysis or experimentation; it also enables us to extract trends from data that would otherwise remain disorganized and incoherent.

We will also discuss modeling. For example, how do we correctly perform tests on the drag on a 3/8-scale model of an automobile in a wind tunnel to predict what the drag would be on the full-size automobile at the same speed? *Must* we use the same speed for model and full-size automobile? How do we scale up the measured model drag to find the automobile drag?

7.1 Nondimensionalizing the Basic Differential Equations

Before describing dimensional analysis let us see what we can learn from our previous analytical descriptions of fluid flow. Consider, for example, a steady incompressible

two-dimensional flow of a Newtonian fluid with constant viscosity (already quite a list of assumptions!). The mass conservation equation (Eq. 5.1c) becomes

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{7.1}$$

and the Navier–Stokes equations (Eqs. 5.27) reduce to

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{7.2}$$

and

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}\right) = -\rho g - \frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{7.3}$$

As we discussed in Section 5.4, these equations form a set of coupled nonlinear partial differential equations for u, v, and p, and are difficult to solve for most flows. Equation 7.1 has dimensions of 1/time, and Eqs. 7.2 and 7.3 have dimensions of force/volume. Let us see what happens when we convert them into dimensionless equations. (Even if you did not study Section 5.4 you will be able to understand the following material.)

To nondimensionalize these equations, divide all lengths by a reference length, L, and all velocities by a reference speed, V_∞, which usually is taken as the freestream velocity. Make the pressure nondimensional by dividing by ρV_∞^2 (twice the freestream dynamic pressure). Denoting nondimensional quantities with asterisks, we obtain

$$x^* = \frac{x}{L}, \quad y^* = \frac{y}{L}, \quad u^* = \frac{u}{V_\infty}, \; v^* = \frac{v}{V_\infty}, \quad \text{and} \quad p^* = \frac{p}{\rho V_\infty^2} \tag{7.4}$$

so that $x = x^*L$, $y = y^*L$, $u = u^*V_\infty$, and so on. We can then substitute into Eqs. 7.1 through 7.3; below we show two representative substitutions:

$$u\frac{\partial u}{\partial x} = u^*V_\infty \frac{\partial(u^*V_\infty)}{\partial(x^*L)} = \frac{V_\infty^2}{L}u^*\frac{\partial u^*}{\partial x^*}$$

and

$$\frac{\partial^2 u}{\partial x^2} = \frac{\partial(u^*V_\infty)}{\partial(x^*L)^2} = \frac{V_\infty}{L^2}\frac{\partial^2 u^*}{\partial x^{*2}}$$

Using this procedure, the equations become

$$\frac{V_\infty}{L}\frac{\partial u^*}{\partial x^*} + \frac{V_\infty}{L}\frac{\partial v^*}{\partial y^*} = 0 \tag{7.5}$$

$$\frac{\rho V_\infty^2}{L}\left(u^*\frac{\partial u^*}{\partial x^*} + v^*\frac{\partial u^*}{\partial y^*}\right) = -\frac{\rho V_\infty^2}{L}\frac{\partial p^*}{\partial x^*} + \frac{\mu V_\infty}{L^2}\left(\frac{\partial^2 u^*}{\partial x^{*2}} + \frac{\partial^2 u^*}{\partial y^{*2}}\right) \tag{7.6}$$

$$\frac{\rho V_\infty^2}{L}\left(u^*\frac{\partial v^*}{\partial x^*} + v^*\frac{\partial v^*}{\partial y^*}\right) = -\rho g - \frac{\rho V_\infty^2}{L}\frac{\partial p^*}{\partial y^*} + \frac{\mu V_\infty}{L^2}\left(\frac{\partial^2 v^*}{\partial x^{*2}} + \frac{\partial^2 v^*}{\partial y^{*2}}\right) \tag{7.7}$$

Dividing Eq. 7.5 by V_∞/L and Eqs. 7.6 and 7.7 by $\rho V_\infty^2/L$ gives

$$\frac{\partial u^*}{\partial x^*} + \frac{\partial v^*}{\partial y^*} = 0 \tag{7.8}$$

$$u^*\frac{\partial u^*}{\partial x^*} + v^*\frac{\partial u^*}{\partial y^*} = -\frac{\partial p^*}{\partial x^*} + \frac{\mu}{\rho V_\infty L}\left(\frac{\partial^2 u^*}{\partial x^{*2}} + \frac{\partial^2 u^*}{\partial y^{*2}}\right) \tag{7.9}$$

$$u^*\frac{\partial v^*}{\partial x^*} + v^*\frac{\partial v^*}{\partial y^*} = -\frac{gL}{V_\infty^2} - \frac{\partial p^*}{\partial y^*} + \frac{\mu}{\rho V_\infty L}\left(\frac{\partial^2 v^*}{\partial x^{*2}} + \frac{\partial^2 v^*}{\partial y^{*2}}\right) \tag{7.10}$$

Equations 7.8, 7.9, and 7.10 are the nondimensional forms of our original equations (Eqs. 7.1, 7.2, 7.3). As such, we can think about their solution (with appropriate boundary conditions) as an exercise in applied mathematics. Equation 7.9 contains a dimensionless coefficient $\mu/\rho V_\infty L$ (which we recognize as the inverse of the Reynolds number) in front of the second-order (viscous) terms; Eq. 7.10 contains this and another dimensionless coefficient, gL/V_∞^2 (which we will discuss shortly) for the gravity force term. We recall from the theory of differential equations that the mathematical form of the solution of such equations is very sensitive to the values of the coefficients in the equations (e.g., certain second-order partial differential equations can be elliptical, parabolic, or hyperbolic depending on coefficient values).

These equations tell us that the solution, and hence the actual flow pattern they describe, depends on the values of the two coefficients. For example, if $\mu/\rho V_\infty L$ is very small (i.e., we have a high Reynolds number), the second-order differentials, representing viscous forces, can be neglected, at least in most of the flow, and we end up with a form of Euler's equations (Eqs. 6.2). We say "in most of the flow" because we have already learned that in reality for this case we will have a boundary layer in which there *is* significant effect of viscosity; in addition, from a mathematical point of view, it is always dangerous to neglect higher-order derivatives, even if their coefficients are small, because reduction to a lower-order equation means we lose a boundary condition (specifically the no-slip condition). We can predict that if $\mu/\rho V_\infty L$ is large or small, then viscous forces will be significant or not, respectively; if gLV_∞^2 is large or small, we can predict that gravity forces will be significant or not, respectively. We can thus gain insight even before attempting a solution to the differential equations. Note that for completeness, we would have to apply the same nondimensionalizing approach to the boundary conditions of the problem, which often introduce further dimensionless coefficients.

Writing nondimensional forms of the governing equations, then, can yield insight into the underlying physical phenomena, and indicate which forces are dominant. If we had two geometrically similar but different scale flows satisfying Eqs. 7.8, 7.9, and 7.10 (for example, a model and a prototype), the equations would only yield the same mathematical results if the two flows had the same values for the two coefficients (i.e., had the same relative importance of gravity, viscous, and inertia forces). This nondimensional form of the equations is also the starting point in numerical methods, which is very often the only way of obtaining their solution. Additional derivations and examples of establishing similitude from the governing equations of a problem are presented in Kline [1] and Hansen [2].

We will now see how the method of dimensional analysis can be used instead of the above procedure to find appropriate dimensionless groupings of physical parameters. As we have mentioned, using dimensionless groupings is very useful for experimental measurements, and we will see in the next two sections that we can obtain them even when we do not have the governing equations such as Eqs. 7.1, 7.2, and 7.3 to work from.

7.2 Nature of Dimensional Analysis

Most phenomena in fluid mechanics depend in a complex way on geometric and flow parameters. For example, consider the drag force on a stationary smooth sphere immersed in a uniform stream. What experiments must be conducted to determine the

drag force on the sphere? To answer this question, we must specify what we believe are the parameters that are important in determining the drag force. Clearly, we would expect the drag force to depend on the size of the sphere (characterized by the diameter, D), the fluid speed, V, and the fluid viscosity, μ. In addition, the density of the fluid, ρ, also might be important. Representing the drag force by F, we can write the symbolic equation

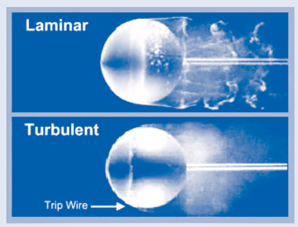
$$F = f(D, V, \rho, \mu)$$

Although we may have neglected parameters on which the drag force depends, such as surface roughness (or may have included parameters on which it does not depend), we have set up the problem of determining the drag force for a stationary sphere in terms of quantities that are both controllable and measurable in the laboratory.

We could set up an experimental procedure for finding the dependence of F on V, D, ρ, and μ. To see how the drag, F, is affected by fluid speed, V, we could place a sphere in a wind tunnel and measure F for a range of V values. We could then run more tests in which we explore the effect on F of sphere diameter, D, by using different diameter spheres. We are already generating a lot of data: If we ran the wind tunnel at, say, 10 different speeds, for 10 different sphere sizes, we'd have 100 data points. We could present these results on one graph (e.g., we could plot 10 curves of F vs. V, one for each sphere size), but acquiring the data would already be time consuming: If we assume each run takes $\frac{1}{2}$ hour, we have already accumulated 50 hours of work! We still wouldn't be finished—we would have to book time using, say, a water tank, where we could repeat all these runs for a different value of ρ and of μ. In principle, we would next have to search out a way to use other fluids to be able to do experiments for a range of ρ and μ values (say, 10 of each). At the end of the day (actually, at the end of about $2\frac{1}{2}$ years of 40-hour weeks!) we would have performed about 10^4 tests. Then we would have to try and make sense of the data: How do we plot, say, curves of F vs. V, with D, ρ, and μ all being parameters? This is a daunting task, even for such a seemingly simple phenomenon as the drag on a sphere!

VIDEO

Flow Around a Sphere 1.



Fortunately we do not have to do all this work. As we will see in Example 7.1, using dimensional analysis, all the data for drag on a smooth sphere can be plotted as a single relationship between two nondimensional parameters in the form

$$\frac{F}{\rho V^2 D^2} = f\left(\frac{\rho V D}{\mu}\right)$$

The form of the function f still must be determined experimentally, but the point is that all spheres, in all fluids, for most velocities will fall on the same curve. Rather than needing to conduct 10^4 experiments, we could establish the nature of the function as accurately with only about 10 tests. The time saved in performing only 10 rather than 10^4 tests is obvious. Even more important is the greater experimental convenience. No longer must we find fluids with 10 different values of density and viscosity. Nor must we make 10 spheres of different diameters. Instead, only the parameter $\rho VD/\mu$ must be varied. This can be accomplished simply by using *one* sphere (e.g., 1 in. diameter), in *one* fluid (e.g., air), and only changing the speed, for example.

VIDEO

Flow Around a Sphere 2.

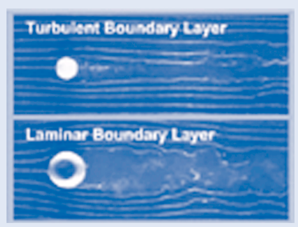


Figure 7.1 shows some classic data for flow over a sphere (the factors $\frac{1}{2}$ and $\pi/4$ have been added to the denominator of the parameter on the left to make it take the form of a commonly used nondimensional group, the drag coefficient, C_D, that we will discuss in detail in Chapter 9). If we performed the experiments as outlined above, our results would fall on the same curve, within experimental error. The data points represent results obtained by various workers for several different fluids and spheres. Note that we end up with a curve that can be used to obtain the drag force on a very wide range of sphere/fluid combinations. For example, it could be used to obtain the drag on a hot-air balloon due to a crosswind, or on a red blood cell (assuming it could be modeled as a

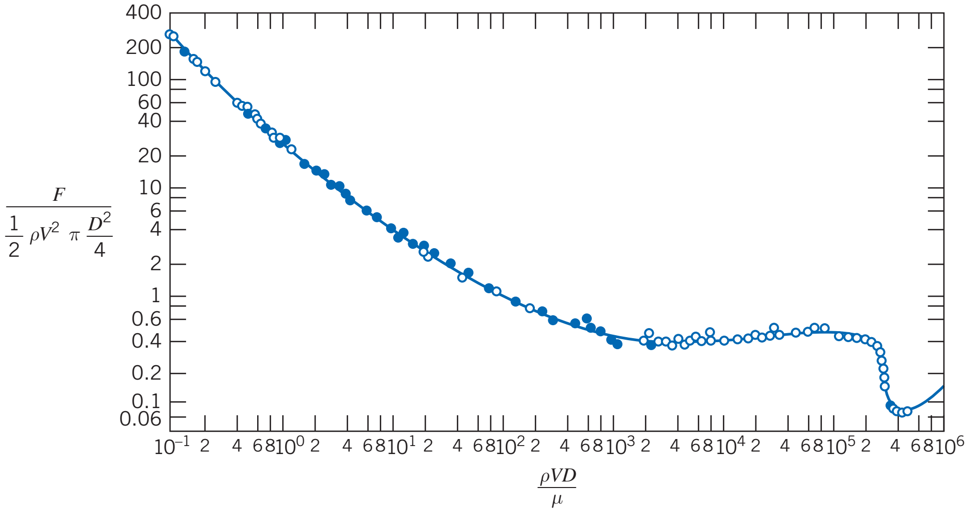


Fig. 7.1 Experimentally derived relation between the nondimensional parameters [3].

sphere) as it moves through the aorta—in either case, given the fluid (ρ and μ), the flow speed V, and the sphere diameter D, we could compute a value for $\rho VD/\mu$, then read the corresponding value for C_D, and finally compute the drag force F.

In Section 7.3 we introduce the *Buckingham Pi* theorem, a formalized procedure for deducing the dimensionless groups appropriate for a given fluid mechanics or other engineering problem. This section, and Section 7.4, may seem a bit difficult to follow; we suggest you read them once, then study Examples 7.1, 7.2, and 7.3 to see how practical and useful the method in fact is, before returning to the two sections for a reread.

The Buckingham Pi theorem is a statement of the relation between a function expressed in terms of dimensional parameters and a related function expressed in terms of nondimensional parameters. The Buckingham Pi theorem allows us to develop the important nondimensional parameters quickly and easily.

7.3 Buckingham Pi Theorem

In the previous section we discussed how the drag F on a sphere depends on the sphere diameter D, fluid density ρ and viscosity μ, and fluid speed V, or

$$F = F(D, \rho, \mu, V)$$

with theory or experiment being needed to determine the nature of function f. More formally, we write

$$g(F, D, \rho, \mu, V) = 0$$

where g is an unspecified function, different from f. The Buckingham Pi theorem [4] states that we can transform a relationship between n parameters of the form

$$g(q_1, q_2, \ldots, q_n) = 0$$

into a corresponding relationship between $n-m$ independent dimensionless Π parameters in the form

$$G(\Pi_1, \Pi_2, \ldots, \Pi_{n-m}) = 0$$

or

$$\Pi_1 = G_1(\Pi_2, \ldots, \Pi_{n-m})$$

where m is *usually* the minimum number, r, of independent dimensions (e.g., mass, length, time) required to define the dimensions of all the parameters $q_1, q_2, \ldots, q_n$. (Sometimes $m \neq r$; we will see this in Example 7.3.) For example, for the sphere problem, we will see (in Example 7.1) that

$$g(F, D, \rho, \mu, V) = 0 \qquad \text{or} \qquad F = F(D, \rho, \mu, V)$$

leads to

$$G\left(\frac{F}{\rho V^2 D^2}, \frac{\mu}{\rho V D}\right) = 0 \qquad \text{or} \qquad \frac{F}{\rho V^2 D^2} = G_1\left(\frac{\mu}{\rho V D}\right)$$

The theorem does not predict the functional form of G or G_1. The functional relation among the independent dimensionless Π parameters must be determined experimentally.

The $n - m$ dimensionless Π parameters obtained from the procedure are independent. A Π parameter is not independent if it can be formed from any combination of one or more of the other Π parameters. For example, if

$$\Pi_5 = \frac{2\Pi_1}{\Pi_2 \Pi_3} \qquad \text{or} \qquad \Pi_6 = \frac{\Pi_1^{3/4}}{\Pi_3^2}$$

then neither Π_5 nor Π_6 is independent of the other dimensionless parameters.

Several methods for determining the dimensionless parameters are available. A detailed procedure is presented in the next section.

Determining the Π Groups 7.4

Regardless of the method to be used to determine the dimensionless parameters, one begins by listing all dimensional parameters that are known (or believed) to affect the given flow phenomenon. Some experience admittedly is helpful in compiling the list. Students, who do not have this experience, often are troubled by the need to apply engineering judgment in an apparent massive dose. However, it is difficult to go wrong if a generous selection of parameters is made.

If you suspect that a phenomenon depends on a given parameter, include it. If your suspicion is correct, experiments will show that the parameter must be included to get consistent results. If the parameter is extraneous, an extra Π parameter may result, but experiments will later show that it may be eliminated. Therefore, do not be afraid to include *all* the parameters that you feel are important.

The six steps listed below (which may seem a bit abstract but are actually easy to do) outline a recommended procedure for determining the Π parameters:

Step 1. *List all the dimensional parameters involved*. (Let n be the number of parameters.) If all of the pertinent parameters are not included, a relation may be obtained, but it will not give the complete story. If parameters that actually have no effect on the physical phenomenon are included, either the process of dimensional analysis will show that these do not enter the relation sought, or one or more dimensionless groups will be obtained that experiments will show to be extraneous.

Step 2. *Select a set of fundamental (primary) dimensions, e.g., MLt or FLt*. (Note that for heat transfer problems you may also need T for temperature, and in electrical systems, q for charge.)

Step 3. *List the dimensions of all parameters in terms of primary dimensions*. (Let r be the number of primary dimensions.) Either force or mass may be selected as a primary dimension.

Step 4. *Select a set of r dimensional parameters that includes all the primary dimensions*. These parameters will all be combined with each of the remaining parameters, one of those at a time, and so will be called repeating parameters. No repeating parameter should have dimensions that are a power of the dimensions of another repeating parameter; for example, do not include both an area (L^2) and a second moment of area (L^4) as repeating parameters. The repeating parameters chosen may appear in all the dimensionless groups obtained; consequently, do *not* include the dependent parameter among those selected in this step.

Step 5. *Set up dimensional equations, combining the parameters selected in Step 4 with each of the other parameters in turn, to form dimensionless groups*. (There will be $n-m$ equations.) Solve the dimensional equations to obtain the $n-m$ dimensionless groups.

Step 6. *Check to see that each group obtained is dimensionless*. If mass was initially selected as a primary dimension, it is wise to check the groups using force as a primary dimension, or vice versa.

The functional relationship among the Π parameters must be determined experimentally. The detailed procedure for determining the dimensionless Π parameters is illustrated in Examples 7.1 and 7.2.

Example 7.1 DRAG FORCE ON A SMOOTH SPHERE

As noted in Section 7.2, the drag force, F, on a smooth sphere depends on the relative speed, V, the sphere diameter, D, the fluid density, ρ, and the fluid viscosity, μ. Obtain a set of dimensionless groups that can be used to correlate experimental data.

Given: $F = f(\rho, V, D, \mu)$ for a smooth sphere.

Find: An appropriate set of dimensionless groups.

Solution:

(Circled numbers refer to steps in the procedure for determining dimensionless Π parameters.)

1 $F \quad V \quad D \quad \rho \quad \mu$ $\qquad n = 5$ dimensional parameters

2 Select primary dimensions M, L, and t.

3 $F \quad V \quad D \quad \rho \quad \mu$

$$\frac{ML}{t^2} \quad \frac{L}{t} \quad L \quad \frac{M}{L^3} \quad \frac{M}{Lt}$$ $\qquad r = 3$ primary dimensions

4 Select repeating parameters ρ, V, D. $\qquad m = r = 3$ repeating parameters

5 Then $n - m = 2$ dimensionless groups will result. Setting up dimensional equations, we obtain

$$\Pi_1 = \rho^a V^b D^c F \quad \text{and} \quad \left(\frac{M}{L^3}\right)^a \left(\frac{L}{t}\right)^b (L)^c \left(\frac{ML}{t^2}\right) = M^0 L^0 t^0$$

Equating the exponents of M, L, and t results in

$$\left.\begin{array}{rrr} M: & a + 1 = 0 & a = -1 \\ L: & -3a + b + c + 1 = 0 & c = -2 \\ t: & -b - 2 = 0 & b = -2 \end{array}\right\} \quad \text{Therefore,} \quad \Pi_1 = \frac{F}{\rho V^2 D^2}$$

Similarly,

$$\Pi_2 = \rho^d V^e D^f \mu \quad \text{and} \quad \left(\frac{M}{L^3}\right)^d \left(\frac{L}{t}\right)^e (L)^f \left(\frac{M}{Lt}\right) = M^0 L^0 t^0$$

$$\left.\begin{array}{rrr} M: & d+1=0 & d=-1 \\ L: & -3d+e+f-1=0 & f=-1 \\ t: & -e-1=0 & e=-1 \end{array}\right\} \quad \text{Therefore,} \quad \Pi_2 = \frac{\mu}{\rho VD}$$

6 Check using F, L, t dimensions

$$[\Pi_1] = \left[\frac{F}{\rho V^2 D^2}\right] \quad \text{and} \quad F\frac{L^4}{Ft^2}\left(\frac{t}{L}\right)^2 \frac{1}{L^2} = 1$$

where [ ] means "has dimensions of," and

$$[\Pi_2] = \left[\frac{\mu}{\rho VD}\right] \quad \text{and} \quad \frac{Ft}{L^2}\,\frac{L^4}{Ft^2}\,\frac{t}{L}\,\frac{1}{L} = 1$$

The functional relationship is $\Pi_1 = f(\Pi_2)$, or

$$\frac{F}{\rho V^2 D^2} = f\left(\frac{\mu}{\rho VD}\right)$$

as noted before. The form of the function, f, must be determined experimentally (see Fig. 7.1).

The *Excel* workbook for this Example is convenient for computing the values of a, b, and c for this and other problems.

Example 7.2 PRESSURE DROP IN PIPE FLOW

The pressure drop, Δp, for steady, incompressible viscous flow through a straight horizontal pipe depends on the pipe length, l, the average velocity, $\overline{V}$, the fluid viscosity, μ, the pipe diameter, D, the fluid density, ρ, and the average "roughness" height, e. Determine a set of dimensionless groups that can be used to correlate data.

Given: $\Delta p = f(\rho, \overline{V}, D, l, \mu, e)$ for flow in a circular pipe.

Find: A suitable set of dimensionless groups.

Solution:

(Circled numbers refer to steps in the procedure for determining dimensionless Π parameters.)

1 $\Delta p \quad \rho \quad \mu \quad \overline{V} \quad l \quad D \quad e \qquad n = 7$ dimensional parameters

2 Choose primary dimensions M, L, and t.

3 $\Delta p \quad \rho \quad \mu \quad \overline{V} \quad l \quad D \quad e$

$$\frac{M}{Lt^2} \quad \frac{M}{L^3} \quad \frac{M}{Lt} \quad \frac{L}{t} \quad L \quad L \quad L \qquad r = 3 \text{ primary dimensions}$$

4 Select repeating parameters ρ, $\overline{V}$, D. $\qquad m = r = 3$ repeating parameters

⑤ Then $n - m = 4$ dimensionless groups will result. Setting up dimensional equations we have:

$$\Pi_1 = \rho^a \overline{V}^b D^c \Delta p \quad \text{and}$$

$$\left(\frac{M}{L^3}\right)^a \left(\frac{L}{t}\right)^b (L)^c \left(\frac{M}{Lt^2}\right) = M^0 L^0 t^0$$

$$\left.\begin{array}{rl} M: & 0 = a + 1 \\ L: & 0 = -3a + b + c - 1 \\ t: & 0 = -b - 2 \end{array}\right\} \quad \begin{array}{l} a = -1 \\ b = -2 \\ c = 0 \end{array}$$

Therefore, $\Pi_1 = \rho^{-1} \overline{V}^{-2} D^0 \Delta p = \dfrac{\Delta p}{\rho \overline{V}^2}$

$$\Pi_3 = \rho^g \overline{V}^h D^i l \quad \text{and}$$

$$\left(\frac{M}{L^3}\right)^g \left(\frac{L}{t}\right)^h (L)^i L = M^0 L^0 t^0$$

$$\left.\begin{array}{rl} M: & 0 = g \\ L: & 0 = -3g + h + i + 1 \\ t: & 0 = -h \end{array}\right\} \quad \begin{array}{l} g = 0 \\ h = 0 \\ i = -1 \end{array}$$

Therefore, $\Pi_3 = \dfrac{l}{D}$

$$\Pi_2 = \rho^d \overline{V}^e D^f \mu \quad \text{and}$$

$$\left(\frac{M}{L^3}\right)^d \left(\frac{L}{t}\right)^e (L)^f \frac{M}{Lt} = M^0 L^0 t^0$$

$$\left.\begin{array}{rl} M: & 0 = d + 1 \\ L: & 0 = -3d + e + f - 1 \\ t: & 0 = -e - 1 \end{array}\right\} \quad \begin{array}{l} d = -1 \\ e = -1 \\ f = -1 \end{array}$$

Therefore, $\Pi_2 = \dfrac{\mu}{\rho \overline{V} D}$

$$\Pi_4 = \rho^j \overline{V}^k D^l e \quad \text{and}$$

$$\left(\frac{M}{L^3}\right)^j \left(\frac{L}{t}\right)^k (L)^l L = M^0 L^0 t^0$$

$$\left.\begin{array}{rl} M: & 0 = j \\ L: & 0 = -3j + k + l + 1 \\ t: & 0 = -k \end{array}\right\} \quad \begin{array}{l} j = 0 \\ k = 0 \\ l = -1 \end{array}$$

Therefore, $\Pi_4 = \dfrac{e}{D}$

⑥ Check, using F, L, t dimensions

$$[\Pi_1] = \left[\frac{\Delta p}{\rho \overline{V}^2}\right] \quad \text{and} \quad \frac{F}{L^2}\frac{L^4}{Ft^2}\frac{t^2}{L^2} = 1 \qquad [\Pi_3] = \left[\frac{l}{D}\right] \quad \text{and} \quad \frac{L}{L} = 1$$

$$[\Pi_2] = \left[\frac{\mu}{\rho \overline{V} D}\right] \quad \text{and} \quad \frac{Ft}{L^2}\frac{L^4}{Ft^2}\frac{t}{L}\frac{1}{L} = 1 \qquad [\Pi_4] = \left[\frac{e}{D}\right] \quad \text{and} \quad \frac{L}{L} = 1$$

Finally, the functional relationship is

$$\Pi_1 = f(\Pi_2, \ \Pi_3, \ \Pi_4)$$

or

$$\frac{\Delta p}{\rho \overline{V}^2} = f\left(\frac{\mu}{\rho \overline{V} D}, \frac{l}{D}, \frac{e}{D}\right)$$

Notes:

✓ As we shall see when we study pipe flow in detail in Chapter 8, this relationship correlates the data well.

✓ Each Π group is unique (e.g., there is only *one* possible dimensionless grouping of μ, ρ, $\overline{V}$, and D).

✓ We can often deduce Π groups by inspection, e.g., l/D is the obvious unique grouping of l with ρ, $\overline{V}$, and D.

The *Excel* workbook for Example 7.1 is convenient for computing the values of a, b, and c for this problem.

The procedure outlined above, where m is taken equal to r (the fewest independent dimensions required to specify the dimensions of all parameters involved), almost always produces the correct number of dimensionless Π parameters. In a few cases, trouble arises because the number of primary dimensions differs when variables are expressed in terms of different systems of dimensions (e.g., MLt or FLt). The value of m can be established with certainty by determining the rank of the dimensional matrix; that rank is m. Although not needed in most applications, for completeness, this procedure is illustrated in Example 7.3.

The $n - m$ dimensionless groups obtained from the procedure are independent but not unique. If a different set of repeating parameters is chosen, different groups result. The repeating parameters are so named because they may appear in all the dimensionless groups obtained. Based on experience, viscosity should appear in only one dimensionless parameter. Therefore μ should *not* be chosen as a repeating parameter.

When we have a choice, it usually works out best to choose density ρ (dimensions M/L^3 in the MLt system), speed V (dimensions L/t), and characteristic length L (dimension L) as repeating parameters because experience shows this generally leads to a set of dimensionless parameters that are suitable for correlating a wide range of experimental data; in addition, ρ, V, and L are usually fairly easy to measure or otherwise obtain. The values of the dimensionless parameters obtained using these repeating parameters almost always have a very tangible meaning, telling you the relative strength of various fluid forces (e.g., viscous) to inertia forces—we will discuss several "classic" ones shortly.

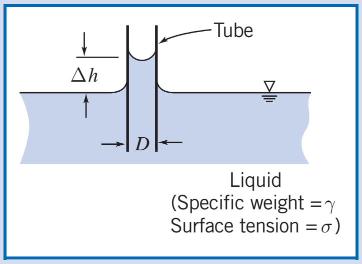
It's also worth stressing that, given the parameters you're combining, *we can often determine the unique dimensional parameters by inspection*. For example, if we had repeating parameters ρ, V, and L and were combining them with a parameter A_f, representing the frontal area of an object, it's fairly obvious that only the combination A_f/L^2 is dimensionless; experienced fluid mechanicians also know that ρV^2 produces dimensions of stress, so any time a stress or force parameter arises, dividing by ρV^2 or $\rho V^2 L^2$ will produce a dimensionless quantity.

We will find useful a measure of the magnitude of fluid inertia forces, obtained from Newton's second law, $F = ma$; the dimensions of inertia force are thus MLt^{-2}. Using ρ, V, and L to build the dimensions of ma leads to the unique combination $\rho V^2 L^2$ (only ρ has dimension M, and only V^2 will produce dimension t^{-2}; L^2 is then required to leave us with MLt^{-2}).

If $n - m = 1$, then a single dimensionless Π parameter is obtained. In this case, the Buckingham Pi theorem indicates that the single Π parameter must be a constant.

Example 7.3 CAPILLARY EFFECT: USE OF DIMENSIONAL MATRIX

When a small tube is dipped into a pool of liquid, surface tension causes a meniscus to form at the free surface, which is elevated or depressed depending on the contact angle at the liquid-solid-gas interface. Experiments indicate that the magnitude of this capillary effect, Δh, is a function of the tube diameter, D, liquid specific weight, γ, and surface tension, σ. Determine the number of independent Π parameters that can be formed and obtain a set.



Given: $\Delta h = f(D, \gamma, \sigma)$

Find: (a) Number of independent Π parameters.
(b) One set of Π parameters.

Solution:
(Circled numbers refer to steps in the procedure for determining dimensionless Π parameters.)

1. $\Delta h \quad D \quad \gamma \quad \sigma \qquad n = 4$ dimensional parameters
2. Choose primary dimensions (use both M, L, t and F, L, t dimensions to illustrate the problem in determining m).

③ (a) M, L, t

$$\begin{array}{cccc} \Delta h & D & \gamma & \sigma \\ L & L & \dfrac{M}{L^2t^2} & \dfrac{M}{t^2} \end{array}$$

$r = 3$ primary dimensions

(b) F, L, t

$$\begin{array}{cccc} \Delta h & D & \gamma & \sigma \\ L & L & \dfrac{F}{L^3} & \dfrac{F}{L} \end{array}$$

$r = 2$ primary dimensions

Thus for each set of primary dimensions we ask, "Is m equal to r?" Let us check each dimensional matrix to find out. The dimensional matrices are

	Δh	D	γ	σ
M	0	0	1	1
L	1	1	−2	0
t	0	0	−2	−2

	Δh	D	γ	σ
F	0	0	1	1
L	1	1	−3	−1

The rank of a matrix is equal to the order of its largest nonzero determinant.

(a)

$$\begin{vmatrix} 0 & 1 & 1 \\ 1 & -2 & 0 \\ 0 & -2 & -2 \end{vmatrix} = 0 - (1)(-2) + (1)(-2) = 0$$

$$\begin{vmatrix} -2 & 0 \\ -2 & -2 \end{vmatrix} = 4 \neq 0 \qquad \therefore\ m = 2,\quad m \neq r$$

④ $m = 2$. Choose D, γ as repeating parameters.

⑤ $n - m = 2$ dimensionless groups will result.

$$\Pi_1 = D^a \gamma^b \Delta h \quad \text{and}$$

$$(L)^a \left(\frac{M}{L^2t^2}\right)^b (L) = M^0L^0t^0$$

$$\left.\begin{array}{rl} M: & b + 0 = 0 \\ L: & a - 2b + 1 = 0 \\ t: & -2b + 0 = 0 \end{array}\right\} \qquad \begin{array}{l} b = 0 \\ a = -1 \end{array}$$

Therefore, $\Pi_1 = \dfrac{\Delta h}{D}$

$$\Pi_2 = D^c \gamma^d \sigma \ \text{and}$$

$$(L)^c \left(\frac{M}{L^2t^2}\right)^d \frac{M}{t^2} = M^0L^0t^0$$

$$\left.\begin{array}{rl} M: & d + 1 = 0 \\ L: & c - 2d = 0 \\ t: & -2d - 2 = 0 \end{array}\right\} \qquad \begin{array}{l} d = -1 \\ c = -2 \end{array}$$

Therefore, $\Pi_2 = \dfrac{\sigma}{D^2\gamma}$

(b)

$$\begin{vmatrix} 1 & 1 \\ -3 & -1 \end{vmatrix} = -1 + 3 = 2 \neq 0$$

$$\therefore\ m = 2,\quad m = r$$

$m = 2$. Choose D, γ as repeating parameters.

$n - m = 2$ dimensionless groups will result.

$$\Pi_1 = D^e \gamma^f \Delta h \quad \text{and}$$

$$(L)^e \left(\frac{F}{L^3}\right)^f L = F^0L^0t^0$$

$$\left.\begin{array}{rl} F: & f = 0 \\ L: & e - 3f + 1 = 0 \end{array}\right\} \qquad e = -1$$

Therefore, $\Pi_1 = \dfrac{\Delta h}{D}$

$$\Pi_2 = D^g \gamma^h \sigma \ \text{and}$$

$$(L)^g \left(\frac{F}{L^3}\right)^h \frac{F}{L} = F^0L^0t^0$$

$$\left.\begin{array}{rl} F: & h + 1 = 0 \\ L: & g - 3h - 1 = 0 \end{array}\right\} \qquad \begin{array}{l} h = -1 \\ g = -2 \end{array}$$

Therefore, $\Pi_2 = \dfrac{\sigma}{D^2\gamma}$

⑥ Check, using F, L, t dimensions

$$[\Pi_1] = \left[\frac{\Delta h}{D}\right] \quad \text{and} \quad \frac{L}{L} = 1$$

$$[\Pi_2] = \left[\frac{\sigma}{D^2\gamma}\right] \quad \text{and} \quad \frac{F}{L}\frac{1}{L^2}\frac{L^3}{F} = 1$$

Check, using M, L, t dimensions

$$[\Pi_1] = \left[\frac{\Delta h}{D}\right] \quad \text{and} \quad \frac{L}{L} = 1$$

$$[\Pi_2] = \left[\frac{\sigma}{D^2\gamma}\right] \quad \text{and} \quad \frac{M}{t^2}\frac{1}{L^2}\frac{L^2t^2}{M} = 1$$

Therefore, both systems of dimensions yield the same dimensionless Π parameters. The predicted functional relationship is

$$\Pi_1 = f(\Pi_2) \qquad \text{or} \qquad \frac{\Delta h}{D} = f\left(\frac{\sigma}{D^2\gamma}\right)$$

Notes:

- ✓ This result is reasonable on physical grounds. The fluid is static; we would not expect time to be an important dimension.
- ✓ We analyzed this problem in Example 2.3, where we found that $\Delta h = 4\sigma\cos(\theta)/\rho g D$ (θ is the contact angle). Hence $\Delta h/D$ is *directly proportional* to $\sigma/D^2\gamma$.
- ✓ The purpose of this problem is to illustrate use of the dimensional matrix to determine the required number of repeating parameters.

7.5 Significant Dimensionless Groups in Fluid Mechanics

Over the years, several hundred different dimensionless groups that are important in engineering have been identified. Following tradition, each such group has been given the name of a prominent scientist or engineer, usually the one who pioneered its use. Several are so fundamental and occur so frequently in fluid mechanics that we should take time to learn their definitions. Understanding their physical significance also gives insight into the phenomena we study.

Forces encountered in flowing fluids include those due to inertia, viscosity, pressure, gravity, surface tension, and compressibility. The ratio of any two forces will be dimensionless. We have previously shown that the inertia force is proportional to $\rho V^2 L^2$.

We can now compare the relative magnitudes of various fluid forces to the inertia force, using the following scheme:

$$\text{Viscous force} \sim \quad \tau A = \mu\frac{du}{dy}A \propto \mu\frac{V}{L}L^2 = \mu VL \quad \text{so} \quad \frac{\text{viscous}}{\text{inertia}} \sim \quad \frac{\mu VL}{\rho V^2L^2} = \frac{\mu}{\rho VL}$$

$$\text{Pressure force} \sim \quad \Delta pA \propto \Delta pL^2 \quad \text{so} \quad \frac{\text{pressure}}{\text{inertia}} \sim \quad \frac{\Delta pL^2}{\rho V^2L^2} = \frac{\Delta p}{\rho V^2}$$

$$\text{Gravity force} \sim \quad mg \propto g\rho L^3 \quad \text{so} \quad \frac{\text{gravity}}{\text{inertia}} \sim \quad \frac{g\rho L^3}{\rho V^2L^2} = \frac{gL}{V^2}$$

Surface tension $\sim$ $\quad \sigma L \quad$ so $\quad \dfrac{\text{surface tension}}{\text{inertia}} \sim \dfrac{\sigma L}{\rho V^2 L^2} = \dfrac{\sigma}{\rho V^2 L}$

Compressibility force $\sim$ $\quad E_v A \propto E_v L^2 \quad$ so $\quad \dfrac{\text{compressibility force}}{\text{inertia}} \sim \dfrac{E_v L^2}{\rho V^2 L^2} = \dfrac{E_v}{\rho V^2}$

All of the dimensionless parameters listed above occur so frequently, and are so powerful in predicting the relative strengths of various fluid forces, that they (slightly modified—usually by taking the inverse) have been given identifying names.

The first parameter, $\mu/\rho V L$, is by tradition inverted to the form $\rho V L/\mu$, and was actually explored independently of dimensional analysis in the 1880s by Osborne Reynolds, the British engineer, who studied the transition between laminar and turbulent flow regimes in a tube. He discovered that the parameter (later named after him)

$$Re = \frac{\rho \overline{V} D}{\mu} = \frac{\overline{V} D}{\nu}$$

is a criterion by which the flow regime may be determined. Later experiments have shown that the *Reynolds number* is a key parameter for other flow cases as well. Thus, in general,

$$Re = \frac{\rho V L}{\mu} = \frac{V L}{\nu} \tag{7.11}$$

where L is a characteristic length descriptive of the flow field geometry. The Reynolds number is the ratio of inertia forces to viscous forces. Flows with "large" Reynolds number generally are turbulent. Flows in which the inertia forces are "small" compared with the viscous forces are characteristically laminar flows.

In aerodynamic and other model testing, it is convenient to modify the second parameter, $\Delta p/\rho V^2$, by inserting a factor $\frac{1}{2}$ to make the denominator represent the dynamic pressure (the factor, of course, does not affect the dimensions). The ratio

$$Eu = \frac{\Delta p}{\frac{1}{2}\rho V^2} \tag{7.12}$$

is formed, where Δp is the local pressure minus the freestream pressure, and ρ and V are properties of the freestream flow. This ratio has been named after Leonhard Euler, the Swiss mathematician who did much early analytical work in fluid mechanics. Euler is credited with being the first to recognize the role of pressure in fluid motion; the Euler equations of Chapter 6 demonstrate this role. The *Euler number* is the ratio of pressure forces to inertia forces. The Euler number is often called the *pressure coefficient*, C_p.

In the study of cavitation phenomena, the pressure difference, Δp, is taken as $\Delta p = p - p_v$, where p is the pressure in the liquid stream, and p_v is the liquid vapor pressure at the test temperature. Combining these with ρ and V in the stream yields the dimensionless parameter called the *cavitation number*,

$$Ca = \frac{p - p_v}{\frac{1}{2}\rho V^2} \tag{7.13}$$

The smaller the cavitation number, the more likely cavitation is to occur. This is usually an unwanted phenomenon.

William Froude was a British naval architect. Together with his son, Robert Edmund Froude, he discovered that the parameter

$$Fr = \frac{V}{\sqrt{gL}} \tag{7.14}$$

was significant for flows with free surface effects. Squaring the *Froude number* gives

$$Fr^2 = \frac{V^2}{gL}$$

which may be interpreted as the ratio of inertia forces to gravity forces (it is the inverse of the third force ratio, V^2/gL, that we discussed above). The length, L, is a characteristic length descriptive of the flow field. In the case of open-channel flow, the characteristic length is the water depth; Froude numbers less than unity indicate subcritical flow and values greater than unity indicate supercritical flow. We will have much more to say on this in Chapter 11.

By convention, the inverse of the fourth force ratio, $\sigma/\rho V^2 L$, discussed above, is called the *Weber* number; it indicates the ratio of inertia to surface tension forces

$$We = \frac{\rho V^2 L}{\sigma} \tag{7.15}$$

The value of the Weber number is indicative of the existence of, and frequency of, capillary waves at a free surface.

In the 1870s, the Austrian physicist Ernst Mach introduced the parameter

$$M = \frac{V}{c} \tag{7.16}$$

where V is the flow speed and c is the local sonic speed. Analysis and experiments have shown that the *Mach number* is a key parameter that characterizes compressibility effects in a flow. The Mach number may be written

$$M = \frac{V}{c} = \frac{V}{\sqrt{\dfrac{dp}{d\rho}}} = \frac{V}{\sqrt{\dfrac{E_v}{\rho}}} \qquad \text{or} \qquad M^2 = \frac{\rho V^2 L^2}{E_v L^2} = \frac{\rho V^2}{E_v}$$

which is the inverse of the final force ratio, $E_v/\rho V^2$, discussed above, and can be interpreted as a ratio of inertia forces to forces due to compressibility. For truly incompressible flow (and note that under some conditions even liquids are quite compressible), $c = \infty$ so that $M = 0$.

Equations 7.11 through 7.16 are some of the most commonly used dimensionless groupings in fluid mechanics because for any flow pattern they immediately (even before performing any experiments or analysis) indicate the relative importance of inertia, viscosity, pressure, gravity, surface tension, and compressibility.

Flow Similarity and Model Studies 7.6

To be useful, a model test must yield data that can be scaled to obtain the forces, moments, and dynamic loads that would exist on the full-scale prototype. What conditions must be met to ensure the similarity of model and prototype flows?

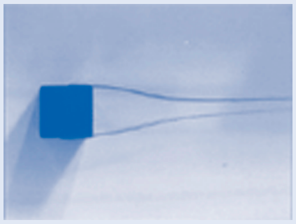
Perhaps the most obvious requirement is that the model and prototype must be geometrically similar. *Geometric similarity* requires that the model and prototype be the same shape, and that all linear dimensions of the model be related to corresponding dimensions of the prototype by a constant scale factor.

A second requirement is that the model and prototype flows must be *kinematically similar*. Two flows are kinematically similar when the velocities at corresponding points are in the same direction and differ only by a constant scale factor. Thus two flows that are kinematically similar also have streamline patterns related by a constant scale factor. Since the boundaries form the bounding streamlines, flows that are kinematically similar must be geometrically similar.

In principle, in order to model the performance in an infinite flow field correctly, kinematic similarity would require that a wind tunnel of infinite cross section be used to obtain data for drag on an object. In practice, this restriction may be relaxed considerably, permitting use of equipment of reasonable size.

VIDEO

Geometric, Not Dynamic, Similarity: Flow Past a Block 1.



Kinematic similarity requires that the regimes of flow be the same for model and prototype. If compressibility or cavitation effects, which may change even the qualitative patterns of flow, are not present in the prototype flow, they must be avoided in the model flow.

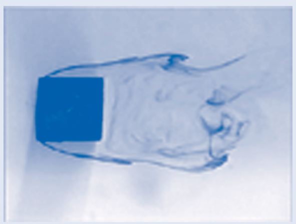
When two flows have force distributions such that identical types of forces are parallel and are related in magnitude by a constant scale factor at all corresponding points, the flows are *dynamically similar*.

The requirements for dynamic similarity are the most restrictive. Kinematic similarity requires geometric similarity; kinematic similarity is a necessary, but not sufficient, requirement for dynamic similarity.

To establish the conditions required for complete dynamic similarity, all forces that are important in the flow situation must be considered. Thus the effects of viscous forces, of pressure forces, of surface tension forces, and so on, must be considered. Test conditions must be established such that all important forces are related by the same scale factor between model and prototype flows. When dynamic similarity exists, data measured in a model flow may be related quantitatively to conditions in the prototype flow. What, then, are the conditions that ensure dynamic similarity between model and prototype flows?

VIDEO

Geometric, Not Dynamic, Similarity: Flow Past a Block 2.



The Buckingham Pi theorem may be used to obtain the governing dimensionless groups for a flow phenomenon; to achieve dynamic similarity between geometrically similar flows, we must make sure that each independent dimensionless group has the same value in the model and in the prototype. Then not only will the forces have the same relative importance, but also the dependent dimensionless group will have the same value in the model and prototype.

For example, in considering the drag force on a sphere in Example 7.1, we began with

$$F = f(D,\ V,\ \rho,\ \mu)$$

The Buckingham Pi theorem predicted the functional relation

$$\frac{F}{\rho V^2 D^2} = f_1\left(\frac{\rho V D}{\mu}\right)$$

In Section 7.5 we showed that the dimensionless parameters can be viewed as ratios of forces. Thus, in considering a model flow and a prototype flow about a sphere (the flows are geometrically similar), the flows also will be dynamically similar if the value of the independent parameter, $\rho VD/\mu$, is duplicated between model and prototype, i.e., if

$$\left(\frac{\rho V D}{\mu}\right)_{\text{model}} = \left(\frac{\rho V D}{\mu}\right)_{\text{prototype}}$$

Furthermore, if

$$Re_{\text{model}} = Re_{\text{prototype}}$$

then the value of the dependent parameter, $F/\rho V^2 D^2$, in the functional relationship, will be duplicated between model and prototype, i.e.,

$$\left(\frac{F}{\rho V^2 D^2}\right)_{\text{model}} = \left(\frac{F}{\rho V^2 D^2}\right)_{\text{prototype}}$$

and the results determined from the model study can be used to predict the drag on the full-scale prototype.

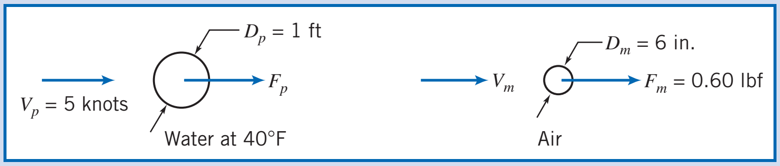
The actual force on the object caused by the fluid is not the same for the model and prototype, but the value of its dimensionless group is. The two tests can be run using different fluids, if desired, as long as the Reynolds numbers are matched. For experimental convenience, test data can be measured in a wind tunnel in air and the results used to predict drag in water, as illustrated in Example 7.4.

Example 7.4 SIMILARITY: DRAG OF A SONAR TRANSDUCER

The drag of a sonar transducer is to be predicted, based on wind tunnel test data. The prototype, a 1-ft diameter sphere, is to be towed at 5 knots (nautical miles per hour) in seawater at 40°F. The model is 6 in. in diameter. Determine the required test speed in air. If the drag of the model at these test conditions is 0.60 lbf, estimate the drag of the prototype.

Given: Sonar transducer to be tested in a wind tunnel.

Find: (a) V_m.
(b) F_p.



Solution:
Since the prototype operates in water and the model test is to be performed in air, useful results can be expected only if cavitation effects are absent in the prototype flow and compressibility effects are absent from the model test. Under these conditions,

$$\frac{F}{\rho V^2 D^2} = f\left(\frac{\rho V D}{\mu}\right)$$

and the test should be run at

$$Re_{\text{model}} = Re_{\text{prototype}}$$

to ensure dynamic similarity. For seawater at 40°F, $\rho = 1.99$ slug/ft^3 and $\nu \approx 1.69 \times 10^{-5}$ ft^2/s. At prototype conditions,

$$V_p = 5\,\frac{\text{nmi}}{\text{hr}} \times 6080\,\frac{\text{ft}}{\text{nmi}} \times \frac{\text{hr}}{3600\,\text{s}} = 8.44 \text{ ft/s}$$

$$Re_p = \frac{V_p D_p}{\nu_p} = 8.44\,\frac{\text{ft}}{\text{s}} \times 1 \text{ ft} \times \frac{\text{s}}{1.69 \times 10^{-5}\ \text{ft}^2} = 4.99 \times 10^5$$

The model test conditions must duplicate this Reynolds number. Thus

$$Re_m = \frac{V_m D_m}{\nu_m} = 4.99 \times 10^5$$

For air at STP, $\rho = 0.00238$ slug/ft^3 and $\nu = 1.57 \times 10^{-4}$ ft^2/s. The wind tunnel must be operated at

$$V_m = Re_m \frac{\nu_m}{D_m} = 4.99 \times 10^5 \times 1.57 \times 10^{-4} \frac{\text{ft}^2}{\text{s}} \times \frac{1}{0.5\,\text{ft}}$$

$$V_m = 157\,\text{ft/s} \longleftarrow V_m$$

This speed is low enough to neglect compressibility effects.

At these test conditions, the model and prototype flows are dynamically similar. Hence

$$\left.\frac{F}{\rho V^2 D^2}\right)_m = \left.\frac{F}{\rho V^2 D^2}\right)_p$$

and

$$F_p = F_m \frac{\rho_p}{\rho_m} \frac{V_p^2}{V_m^2} \frac{D_p^2}{D_m^2} = 0.60\,\text{lbf} \times \frac{1.99}{0.00238} \times \frac{(8.44)^2}{(157)^2} \times \frac{1}{(0.5)^2}$$

$$F_p = 5.8\,\text{lbf} \longleftarrow F_p$$

This problem:

✓ Demonstrates the calculation of prototype values from model test data.

✓ "Reinvented the wheel": the results for drag on a smooth sphere are very well known, so we did not need to do a model experiment but instead could have simply read from the graph of Fig. 7.1 the value of $C_D = F_p / \left(\frac{1}{2}\rho V_p^2 \frac{\pi}{4} D_p^2\right) \approx 0.1$, corresponding to a Reynolds number of 4.99×10^5. Then $F_p \approx 5.6$ lbf can easily be computed. We will have more to say on drag coefficients in Chapter 9.

If cavitation were expected—if the sonar probe were operated at high speed near the free surface of the seawater—then useful results could not be obtained from a model test in air.

Incomplete Similarity

We have shown that to achieve complete dynamic similarity between geometrically similar flows, it is necessary to duplicate the values of the independent dimensionless groups; by so doing the value of the dependent parameter is then duplicated.

In the simplified situation of Example 7.4, duplicating the Reynolds number value between model and prototype ensured dynamically similar flows. Testing in air allowed the Reynolds number to be duplicated exactly (this also could have been accomplished in a water tunnel for this situation). The drag force on a sphere actually depends on the nature of the boundary-layer flow. Therefore, geometric similarity requires that the relative surface roughness of the model and prototype be the same. This means that relative roughness also is a parameter that must be duplicated between model and prototype situations. If we assume that the model was constructed carefully, measured values of drag from model tests could be scaled to predict drag for the operating conditions of the prototype.

In many model studies, to achieve dynamic similarity requires duplication of several dimensionless groups. In some cases, complete dynamic similarity between model and prototype may not be attainable. Determining the drag force (resistance) of a surface ship is an example of such a situation. Resistance on a surface ship arises from skin friction on the hull (viscous forces) and surface wave resistance (gravity forces). Complete dynamic similarity requires that both Reynolds and Froude numbers be duplicated between model and prototype.

In general it is not possible to predict wave resistance analytically, so it must be modeled. This requires that

$$Fr_m = \frac{V_m}{(gL_m)^{1/2}} = Fr_p = \frac{V_p}{(gL_p)^{1/2}}$$

To match Froude numbers between model and prototype therefore requires a velocity ratio of

$$\frac{V_m}{V_p} = \left(\frac{L_m}{L_p}\right)^{1/2}$$

to ensure dynamically similar surface wave patterns.

Hence for any model length scale, matching the Froude numbers determines the velocity ratio. Only the kinematic viscosity can then be varied to match Reynolds numbers. Thus

$$Re_m = \frac{V_m L_m}{\nu_m} = Re_p = \frac{V_p L_p}{\nu_p}$$

leads to the condition that

$$\frac{\nu_m}{\nu_p} = \frac{V_m}{V_p}\frac{L_m}{L_P}$$

If we use the velocity ratio obtained from matching the Froude numbers, equality of Reynolds numbers leads to a kinematic viscosity ratio requirement of

$$\frac{\nu_m}{\nu_p} = \left(\frac{L_m}{L_p}\right)^{1/2}\frac{L_m}{L_p} = \left(\frac{L_m}{L_p}\right)^{3/2}$$

If $L_m/L_p = \frac{1}{100}$ (a typical length scale for ship model tests), then ν_m/ν_p must be $\frac{1}{1000}$. Figure A.3 shows that mercury is the only liquid with kinematic viscosity less than that of water. However, it is only about an order of magnitude less, so the kinematic viscosity ratio required to duplicate Reynolds numbers cannot be attained.

We conclude that we have a problem: it is impossible in practice for this model/prototype scale of $\frac{1}{100}$ to satisfy both the Reynolds number and Froude number criteria; at best we will be able to satisfy only one of them. In addition, water is the only practical fluid for most model tests of free-surface flows. To obtain complete dynamic similarity then would require a full-scale test. However, all is not lost: Model studies do provide useful information even though complete similarity cannot be obtained. As an example, Fig. 7.2 shows data from a test of a 1:80 scale model of a ship conducted at the U.S. Naval Academy Hydromechanics Laboratory. The plot displays "resistance coefficient" data versus Froude number. The square points are calculated from values of total resistance measured in the test. We would like to obtain the corresponding total resistance curve for the full-scale ship.

If you think about it, we can *only* measure the total drag (the square data points). The total drag is due to both wave resistance (dependent on the Froude number) and friction resistance (dependent on the Reynolds number), and it's not possible to determine experimentally how much each contributes. We *cannot* use the total drag curve of Fig. 7.2 for the full-scale ship because, as we have discussed above, we can never set up the model conditions so that its Reynolds number *and* Froude number match those of the full-scale ship. Nevertheless, we would like to extract from Fig. 7.2 the corresponding total drag curve for the full-scale ship. In many experimental situations we need to use a creative "trick" to come up with a solution. In this case, the experimenters used boundary-layer theory (which we discuss in Chapter 9) to *predict* the viscous resistance component of the model (shown as diamonds in Fig. 7.2); then they estimated the wave resistance (not obtainable from theory) by simply subtracting this theoretical viscous resistance from the experimental total resistance, point by point (shown as circles in Fig. 7.2).

Using this clever idea (typical of the kind of experimental and analytical approaches experimentalists need to employ), Fig. 7.2 therefore gives the wave

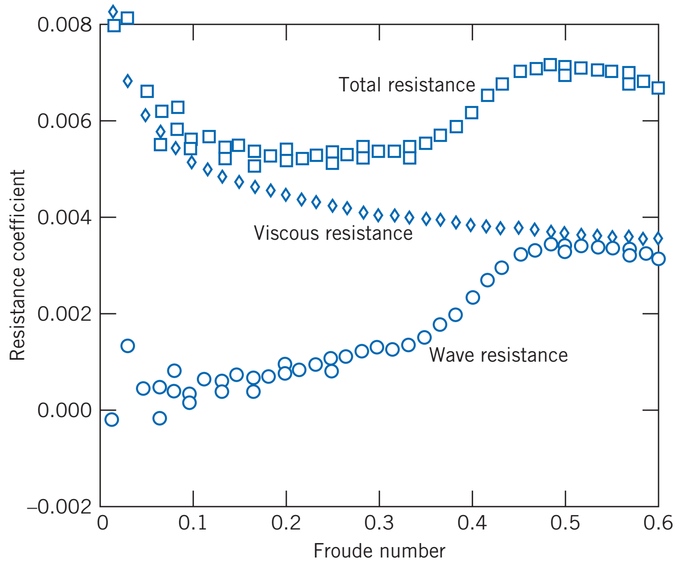


Fig. 7.2 Data from test of 1:80 scale model of U.S. Navy guided missile frigate *Oliver Hazard Perry* (FFG-7). (Data from U.S. Naval Academy Hydromechanics Laboratory, courtesy of Professor Bruce Johnson.)

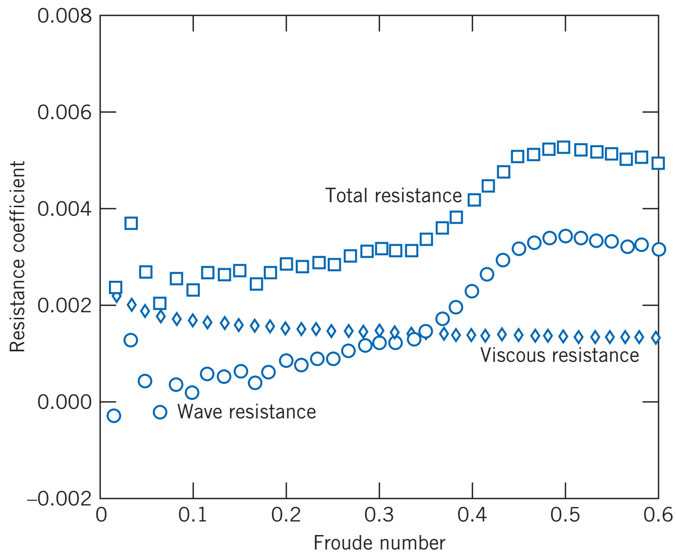


Fig. 7.3 Resistance of full-scale ship predicted from model test results. (Data from U.S. Naval Academy Hydromechanics Laboratory, courtesy of Professor Bruce Johnson.)

resistance of the model as a function of Froude number. It is *also* valid for the full-scale ship, because wave resistance depends only on the Froude number! We can now build a graph similar to Fig. 7.2 valid for the full-scale ship: Simply compute from boundary-layer theory the viscous resistance of the full-scale ship and add this to the wave resistance values, point by point. The result is shown in Fig. 7.3. The wave resistance points are identical to those in Fig. 7.2; the viscous resistance points are computed from theory (and are different from those of Fig. 7.2); and the predicted total resistance curve for the full-scale ship is finally obtained.

In this example, incomplete modeling was overcome by using analytical computations; the model experiments modeled the Froude number, but not the Reynolds number.

Because the Reynolds number cannot be matched for model tests of surface ships, the boundary-layer behavior is not the same for model and prototype. The model Reynolds number is only $(L_m/L_p)^{3/2}$ as large as the prototype value, so the extent of laminar flow in the boundary layer on the model is too large by a corresponding factor. The method just described assumes that boundary-layer behavior can be scaled. To make this possible, the model boundary layer is "tripped" or "stimulated" to become turbulent at a location that corresponds to the behavior on the full-scale vessel. "Studs" were used to stimulate the boundary layer for the model test results shown in Fig. 7.2.

A correction sometimes is added to the full-scale coefficients calculated from model test data. This correction accounts for roughness, waviness, and unevenness that inevitably are more pronounced on the full-scale ship than on the model. Comparisons between predictions from model tests and measurements made in full-scale trials suggest an overall accuracy within ± 5 percent [5].

As we will see in Chapter 11, the Froude number is an important parameter in the modeling of rivers and harbors. In these situations it is not practical to obtain complete similarity. Use of a reasonable model scale would lead to extremely small water depths, so that viscous forces and surface tension forces would have much larger relative effects in the model flow than in the prototype. Consequently, different length scales are used for the vertical and horizontal directions. Viscous forces in the deeper model flow are increased using artificial roughness elements.

Emphasis on fuel economy has made reduction of aerodynamic drag important for automobiles, trucks, and buses. Most work on development of low-drag configurations is done using model tests. Traditionally, automobile models have been built to $\frac{3}{8}$ scale, at which a model of a full-size automobile has a frontal area of about 0.3 m^2. Thus testing can be done in a wind tunnel with test section area of 6 m^2 or larger. At $\frac{3}{8}$ scale, a wind speed of about 150 mph is needed to model a prototype automobile traveling at the legal speed limit. Thus there is no problem with compressibility effects, but the scale models are expensive and time-consuming to build.

A large wind tunnel (test section dimensions are 5.4 m high, 10.4 m wide, and 21.3 m long; maximum air speed is 250 km/hr with the tunnel empty) is used by General Motors to test full-scale automobiles at highway speeds. The large test section allows use of production autos or of full-scale clay mockups of proposed auto body styles. Many other vehicle manufacturers are using comparable facilities; Fig. 7.4 shows a full-size sedan under test in the Volvo wind tunnel. The relatively low speed permits flow visualization using tufts or "smoke" streams.[1] Using full-size "models," stylists and engineers can work together to achieve optimum results.

It is harder to achieve dynamic similarity in tests of trucks and buses; models must be made to smaller scale than those for automobiles.[2] A large scale for truck and bus testing is 1:8. To achieve complete dynamic similarity by matching Reynolds numbers at this scale would require a test speed of 440 mph. This would introduce unwanted compressibility effects, and model and prototype flows would not be kinematically similar. Fortunately, trucks and buses are "bluff" objects. Experiments show that above a certain Reynolds number, their nondimensional drag becomes independent

[1]A mixture of liquid nitrogen and steam may be used to produce "smoke" streaklines that evaporate and do not clog the fine mesh screens used to reduce the turbulence level in a wind tunnel. Streaklines may be made to appear "colored" in photos by placing a filter over the camera lens. This and other techniques for flow visualization are detailed in Reference [6] and Merzkirch [7].

[2]The vehicle length is particularly important in tests at large yaw angles to simulate crosswind behavior. Tunnel blockage considerations limit the acceptable model size. See Reference [8] for recommended practices.

Fig. 7.4 Full-scale automobile under test in Volvo wind tunnel, using smoke streaklines for flow visualization. (Photograph courtesy of Volvo Cars of North America, Inc.)

of Reynolds number [8]. (Figure 7.1 actually shows an example of this—for a sphere, the dimensionless drag is approximately constant for $2000 < Re < 2 \times 10^5$.) Although similarity is not complete, measured test data can be scaled to predict prototype drag forces. The procedure is illustrated in Example 7.5.

Example 7.5 INCOMPLETE SIMILARITY: AERODYNAMIC DRAG ON A BUS

The following wind tunnel test data from a 1:16 scale model of a bus are available:

Air Speed (m/s)	18.0	21.8	26.0	30.1	35.0	38.5	40.9	44.1	46.7
Drag Force (N)	3.10	4.41	6.09	7.97	10.7	12.9	14.7	16.9	18.9

Using the properties of standard air, calculate and plot the dimensionless aerodynamic drag coefficient,

$$C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A}$$

versus Reynolds number $Re = \rho V w/\mu$, where w is model width. Find the minimum test speed above which C_D remains constant. Estimate the aerodynamic drag force and power requirement for the prototype vehicle at 100 km/hr. (The width and frontal area of the prototype are 8 ft and 84 ft^2, respectively.)

Given: Data from a wind tunnel test of a model bus. Prototype dimensions are width of 8 ft and frontal area of 84 ft^2. Model scale is 1:16. Standard air is the test fluid.

Find: (a) Aerodynamic drag coefficient, $C_D = F_D/\frac{1}{2}\rho V^2 A$, versus Reynolds number, $Re = \rho V w/\mu$; plot.
(b) Speed above which C_D is constant.
(c) Estimated aerodynamic drag force and power required for the full-scale vehicle at 100 km/hr.

Solution:

The model width is

$$w_m = \frac{1}{16} w_p = \frac{1}{16} \times 8 \text{ ft} \times 0.3048 \frac{\text{m}}{\text{ft}} = 0.152 \text{ m}$$

The model area is

$$A_m = \left(\frac{1}{16}\right)^2 A_p = \left(\frac{1}{16}\right)^2 \times 84 \text{ ft}^2 \times (0.305)^2 \frac{\text{m}^2}{\text{ft}^2} = 0.0305 \text{ m}^2$$

The aerodynamic drag coefficient may be calculated as

$$C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A}$$

$$= 2 \times F_D(\text{N}) \times \frac{\text{m}^3}{1.23 \text{ kg}} \times \frac{\text{s}^2}{(V)^2 \text{ m}^2} \times \frac{1}{0.0305 \text{ m}^2} \times \frac{\text{kg} \cdot \text{m}}{\text{N} \cdot \text{s}^2}$$

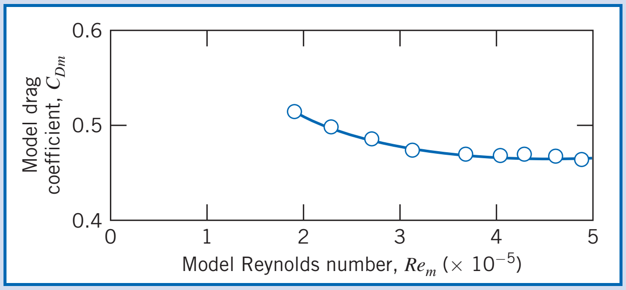
$$C_D = \frac{53.3\, F_D(\text{N})}{[V(\text{m/s})]^2}$$

The Reynolds number may be calculated as

$$Re = \frac{\rho V w}{\mu} = \frac{V w}{\nu} = V \frac{\text{m}}{\text{s}} \times 0.152 \text{ m} \times \frac{\text{s}}{1.46 \times 10^{-5} \text{ m}^2}$$

$$Re = 1.04 \times 10^4\, V(\text{m/s})$$

The calculated values are plotted in the following figure:



C_{D_m} versus Re_m

The plot shows that the model drag coefficient becomes constant at $C_{Dm} \approx 0.46$ above $Re_m = 4 \times 10^5$, which corresponds to an air speed of approximately 40 m/s. Since the drag coefficient is independent of Reynolds number above $Re \approx 4 \times 10^5$, then for the prototype vehicle ($Re \approx 4.5 \times 10^6$), $C_D \approx 0.46$. The drag force on the full-scale vehicle is

$$F_{D_p} = C_D \frac{1}{2} \rho V_p^2 A_p$$

$$= \frac{0.46}{2} \times 1.23 \frac{\text{kg}}{\text{m}^3} \left(100 \frac{\text{km}}{\text{hr}} \times 1000 \frac{\text{m}}{\text{km}} \times \frac{\text{hr}}{3600 \text{ s}}\right)^2 \times 84 \text{ ft}^2 \times (0.305)^2 \frac{\text{m}^2}{\text{ft}^2} \times \frac{\text{N} \cdot \text{s}^2}{\text{kg} \cdot \text{m}}$$

$$F_{D_p} = 1.71 \text{ kN} \quad F_{Dp}$$

The corresponding power required to overcome aerodynamic drag is

$$\mathcal{P}_p = F_{D_p} V_p$$

$$= 1.71 \times 10^3 \text{ N} \times 100 \frac{\text{km}}{\text{hr}} \times 1000 \frac{\text{m}}{\text{km}} \times \frac{\text{hr}}{3600 \text{ s}} \times \frac{\text{W} \cdot \text{s}}{\text{N} \cdot \text{m}}$$

$$\mathcal{P}_p = 47.5 \text{ kW} \longleftarrow \mathcal{P}_p$$

This problem illustrates a common phenomenon in aerodynamics: Above a certain minimum Reynolds number the drag coefficient of an object usually approaches a constant—that is, becomes independent of the Reynolds number. Hence, in these situations we do not have to match the Reynolds numbers of the model and prototype in order for them to have the same drag coefficient—a considerable advantage. However, the SAE *Recommended Practices* [8] suggests $Re \geq 2 \times 10^6$ for truck and bus testing.

For additional details on techniques and applications of dimensional analysis consult [9–12].

Scaling with Multiple Dependent Parameters

In some situations of practical importance there may be more than one dependent parameter. In such cases, dimensionless groups must be formed separately for each dependent parameter.

As an example, consider a typical centrifugal pump. The detailed flow pattern within a pump changes with volume flow rate and speed; these changes affect the pump's performance. Performance parameters of interest include the pressure rise (or head) developed, the power input required, and the machine efficiency measured under specific operating conditions.[3] Performance curves are generated by varying an independent parameter such as the volume flow rate. Thus the independent variables are volume flow rate, angular speed, impeller diameter, and fluid properties. Dependent variables are the several performance quantities of interest.

Finding dimensionless parameters begins from the symbolic equations for the dependence of head, h (energy per unit mass, L^2/t^2), and power, $\mathcal{P}$, on the independent parameters, given by

$$h = g_1(Q, \ \rho, \ \omega, \ D, \ \mu)$$

and

$$\mathcal{P} = g_2(Q, \ \rho, \ \omega, \ D, \ \mu)$$

Straightforward use of the Pi theorem gives the dimensionless *head coefficient* and *power coefficient* as

$$\frac{h}{\omega^2 D^2} = f_1\left(\frac{Q}{\omega D^3}, \ \frac{\rho \omega D^2}{\mu}\right) \tag{7.17}$$

[3]Efficiency is defined as the ratio of power delivered to the fluid divided by input power, $\eta = \mathcal{P}/\mathcal{P}_{\text{in}}$. For incompressible flow, we will see in Chapter 8 that the energy equation reduces to $\mathcal{P} = \rho Q h$ (when "head" h is expressed as energy per unit mass) or to $\mathcal{P} = \rho g Q H$ (when head H is expressed as energy per unit weight).

and

$$\frac{\mathcal{P}}{\rho\omega^3 D^5} = f_2\left(\frac{Q}{\omega D^3}, \frac{\rho\omega D^2}{\mu}\right) \tag{7.18}$$

The dimensionless parameter $Q/\omega D^3$ in these equations is called the *flow coefficient*. The dimensionless parameter $\rho\omega D^2/\mu$ $(\propto \rho VD/\mu)$ is a form of Reynolds number.

Head and power in a pump are developed by inertia forces. Both the flow pattern within a pump and the pump performance change with volume flow rate and speed of rotation. Performance is difficult to predict analytically except at the design point of the pump, so it is measured experimentally. Typical characteristic curves plotted from experimental data for a centrifugal pump tested at constant speed are shown in Fig. 7.5 as functions of volume flow rate. The head, power, and efficiency curves in Fig. 7.5 are smoothed through points calculated from measured data. Maximum efficiency usually occurs at the design point.

Complete similarity in pump performance tests would require identical flow coefficients and Reynolds numbers. In practice, it has been found that viscous effects are relatively unimportant when two geometrically similar machines operate under "similar" flow conditions. Thus, from Eqs. 7.17 and 7.18, when

$$\frac{Q_1}{\omega_1 D_1^3} = \frac{Q_2}{\omega_2 D_2^3} \tag{7.19}$$

it follows that

$$\frac{h_1}{\omega_1^2 D_1^2} = \frac{h_2}{\omega_2^2 D_2^2} \tag{7.20}$$

and

$$\frac{\mathcal{P}_1}{\rho_1 \omega_1^3 D_1^5} = \frac{\mathcal{P}_2}{\rho_2 \omega_2^3 D_2^5} \tag{7.21}$$

The empirical observation that viscous effects are unimportant under similar flow conditions allows use of Eqs. 7.19 through 7.21 to scale the performance characteristics of machines to different operating conditions, as either the speed or diameter is changed. These useful scaling relationships are known as pump or fan "laws." If operating conditions for one machine are known, operating conditions for any geometrically similar machine can be found by changing D and ω according to Eqs. 7.19 through 7.21. (More details on dimensional analysis, design, and performance curves for fluid machinery are presented in Chapter 10.)

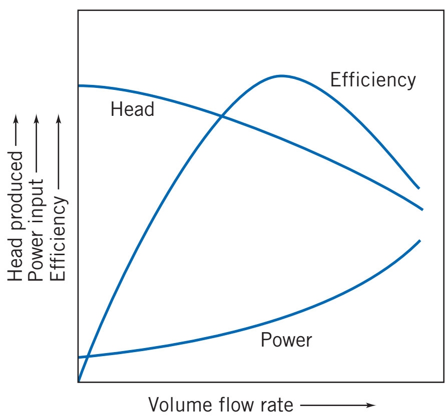


Fig. 7.5 Typical characteristic curves for centrifugal pump tested at constant speed.

Another useful pump parameter can be obtained by eliminating the machine diameter from Eqs. 7.19 and 7.20. If we designate $\Pi_1 = Q/\omega D^3$ and $\Pi_2 = h/\omega^2 D^2$, then the ratio $\Pi_1^{1/2}/\Pi_2^{3/4}$ is another dimensionless parameter; this parameter is the *specific speed, N_s*,

$$N_s = \frac{\omega Q^{1/2}}{h^{3/4}} \tag{7.22a}$$

The specific speed, as defined in Eq. 7.22a, is a dimensionless parameter (provided that the head, h, is expressed as energy per unit mass). You may think of specific speed as the speed required for a machine to produce unit head at unit volume flow rate. A constant specific speed describes all operating conditions of geometrically similar machines with similar flow conditions.

Although specific speed is a dimensionless parameter, it is common practice to use a convenient but inconsistent set of units in specifying the variables ω and Q, and to use the energy per unit weight H in place of energy per unit mass h in Eq. 7.22a. When this is done the specific speed,

$$N_{s_{cu}} = \frac{\omega Q^{1/2}}{H^{3/4}} \tag{7.22b}$$

is not a unitless parameter and its magnitude depends on the units used to calculate it. Customary units used in U.S. engineering practice for pumps are rpm for ω, gpm for Q, and feet (energy per unit weight) for H. In these customary U.S. units, "low" specific speed means $500 < N_{s_{cu}} < 4000$ and "high" means $10{,}000 < N_{s_{cu}} < 15{,}000$. Example 7.6 illustrates use of the pump scaling laws and specific speed parameter. More details of specific speed calculations and additional examples of applications to fluid machinery are presented in Chapter 10.

Example 7.6 PUMP "LAWS"

A centrifugal pump has an efficiency of 80 percent at its design-point specific speed of 2000 (units of rpm, gpm, and feet). The impeller diameter is 8 in. At design-point flow conditions, the volume flow rate is 300 gpm of water at 1170 rpm. To obtain a higher flow rate, the pump is to be fitted with a 1750 rpm motor. Use the pump "laws" to find the design-point performance characteristics of the pump at the higher speed. Show that the specific speed remains constant for the higher operating speed. Determine the motor size required.

Given: Centrifugal pump with design specific speed of 2000 (in rpm, gpm, and feet units). Impeller diameter is $D = 8$ in. At the pump's design-point flow conditions, $\omega = 1170$ rpm and $Q = 300$ gpm, with water.

Find: (a) Performance characteristics,
(b) specific speed, and
(c) motor size required, for similar flow conditions at 1750 rpm.

Solution: From pump "laws," $Q/\omega D^3 =$ constant, so

$$Q_2 = Q_1 \frac{\omega_2}{\omega_1}\left(\frac{D_2}{D_1}\right)^3 = 300 \text{ gpm}\left(\frac{1750}{1170}\right)(1)^3 = 449 \text{ gpm} \longleftarrow Q_2$$

The pump head is not specified at $\omega_1 = 1170$ rpm, but it can be calculated from the specific speed, $N_{s_{cu}} = 2000$. Using the given units and the definition of $N_{s_{cu}}$,

$$N_{s_{cu}} = \frac{\omega Q^{1/2}}{H^{3/4}} \qquad \text{so} \qquad H_1 = \left(\frac{\omega_1 Q_1^{1/2}}{N_{s_{cu}}}\right)^{4/3} = 21.9 \text{ ft}$$

Then $H/\omega^2 D^2 = \text{constant}$, so

$$H_2 = H_1\left(\frac{\omega_2}{\omega_1}\right)^2\left(\frac{D_2}{D_1}\right)^2 = 21.9\text{ ft}\left(\frac{1750}{1170}\right)^2(1)^2 = 49.0\text{ ft} \longleftarrow H_2$$

The pump output power is $\mathcal{P}_1 = \rho g Q_1 H_1$, so at $\omega_1 = 1170$ rpm,

$$\mathcal{P}_1 = 1.94\frac{\text{slug}}{\text{ft}^3}\times 32.2\frac{\text{ft}}{\text{s}^2}\times 300\frac{\text{gal}}{\text{min}}\times 21.9\text{ ft}\times\frac{\text{ft}^3}{7.48\text{ gal}}\times\frac{\text{min}}{60\text{ s}}\times\frac{\text{lbf}\cdot\text{s}^2}{\text{slug}\cdot\text{ft}}\times\frac{\text{hp}\cdot\text{s}}{550\text{ ft}\cdot\text{lbf}}$$

$$\mathcal{P}_1 = 1.66\text{ hp}$$

But $\mathcal{P}/\rho\omega^3 D^5 = \text{constant}$, so

$$\mathcal{P}_2 = \mathcal{P}_1\left(\frac{\rho_2}{\rho_1}\right)\left(\frac{\omega_2}{\omega_1}\right)^3\left(\frac{D_2}{D_1}\right)^5 = 1.66\text{ hp}(1)\left(\frac{1750}{1170}\right)^3(1)^5 = 5.55\text{ hp} \longleftarrow \mathcal{P}_2$$

The required input power may be calculated as

$$\mathcal{P}_{\text{in}} = \frac{\mathcal{P}_2}{\eta} = \frac{5.55\text{ hp}}{0.80} = 6.94\text{ hp} \longleftarrow \mathcal{P}_{\text{in}}$$

Thus a 7.5-hp motor (the next larger standard size) probably would be specified.

The specific speed at $\omega_2 = 1750$ rpm is

$$N_{s_{cu}} = \frac{\omega Q^{1/2}}{H^{3/4}} = \frac{1750(449)^{1/2}}{(49.0)^{3/4}} = 2000 \longleftarrow N_{s_{cu}}$$

This problem illustrates application of the pump "laws" and specific speed to scaling of performance data. Pump and fan "laws" are used widely in industry to scale performance curves for families of machines from a single performance curve, and to specify drive speed and power in machine applications.

Comments on Model Testing

While outlining the procedures involved in model testing, we have tried not to imply that testing is a simple task that automatically gives results that are easily interpreted, accurate, and complete. As in all experimental work, careful planning and execution are needed to obtain valid results. Models must be constructed carefully and accurately, and they must include sufficient detail in areas critical to the phenomenon being measured. Aerodynamic balances or other force measuring systems must be aligned carefully and calibrated correctly. Mounting methods must be devised that offer adequate rigidity and model motion, yet do not interfere with the phenomenon being measured. References [13–15] are considered the standard sources for details of wind tunnel test techniques. More specialized techniques for water impact testing are described in Waugh and Stubstad [16].

Experimental facilities must be designed and constructed carefully. The quality of flow in a wind tunnel must be documented. Flow in the test section should be as nearly uniform as possible (unless the desire is to simulate a special profile such as an atmospheric boundary layer), free from angularity, and with little swirl. If they interfere with measurements, boundary layers on tunnel walls must be removed by suction or energized by blowing. Pressure gradients in a wind tunnel test section may cause erroneous drag-force readings due to pressure variations in the flow direction.

Special facilities are needed for unusual conditions or for special test requirements, especially to achieve large Reynolds numbers. Many facilities are so large or specialized that they cannot be supported by university laboratories or private industry. A few examples include [17–19]:

- National Full-Scale Aerodynamics Complex, NASA, Ames Research Center, Moffett Field, California.
  Two wind tunnel test sections, powered by a 125,000 hp electric drive system:
  - 40 ft high and 80 ft wide (12 × 24 m) test section, maximum wind speed of 300 knots.
  - 80 ft high and 120 ft wide (24 × 36 m) test section, maximum wind speed of 137 knots.
- U.S. Navy, David Taylor Research Center, Carderock, Maryland.
  - High-Speed Towing Basin 2968 ft long, 21 ft wide, and 16 ft deep. Towing carriage can travel at up to 100 knots while measuring drag loads to 8000 lbf and side loads to 2000 lbf.
  - 36 in. variable-pressure water tunnel with 50 knot maximum test speed at pressures between 2 and 60 psia.
  - Anechoic Flow Facility with quiet, low-turbulence air flow in 8 ft square by 21 ft-long open-jet test section. Flow noise at maximum speed of 200 ft/s is less than that of conversational speech.
- U.S. Army Corps of Engineers, Sausalito, California.
  - San Francisco Bay and Delta Model with slightly more than 1 acre in area, 1:1000 horizontal scale and 1:100 vertical scale, 13,500 gpm of pumping capacity, use of fresh and salt water, and tide simulation.
- NASA, Langley Research Center, Hampton, Virginia.
  - National Transonic Facility (NTF) with cryogenic technology (temperatures as low as −300°F) to reduce gas viscosity, raising Reynolds number by a factor of 6, while halving drive power.

7.7 Summary and Useful Equations

In this chapter we have:

✓ Obtained dimensionless coefficients by nondimensionalizing the governing differential equations of a problem.
✓ Stated the Buckingham Pi theorem and used it to determine the independent and dependent dimensionless parameters from the physical parameters of a problem.
✓ Defined a number of important dimensionless groups: the Reynolds number, Euler number, cavitation number, Froude number, Weber number, and Mach number, and discussed their physical significance.

We have also explored some ideas behind modeling: geometric, kinematic, and dynamic similarity, incomplete modeling, and predicting prototype results from model tests.

Note: Most of the Useful Equations in the table below have a number of constraints or limitations—*be sure to refer to their page numbers for details*!

Useful Equations

Reynolds number (inertia to viscous):	$Re = \dfrac{\rho V L}{\mu} = \dfrac{VL}{\nu}$	**(7.11)**	Page 304
Euler number (pressure to inertia):	$Eu = \dfrac{\Delta p}{\frac{1}{2}\rho V^2}$	**(7.12)**	Page 304
Cavitation number:	$Ca = \dfrac{p - p_v}{\frac{1}{2}\rho V^2}$	**(7.13)**	Page 304

Froude number (inertia to gravity):	$Fr = \dfrac{V}{\sqrt{gL}}$	**(7.14)**	Page 305
Weber number (inertia to surface tension):	$We = \dfrac{\rho V^2 L}{\sigma}$	**(7.15)**	Page 305
Mach number (inertia to compressibility):	$M = \dfrac{V}{c}$	**(7.16)**	Page 305
Centrifugal pump specific speed (in terms of head h):	$N_s = \dfrac{\omega Q^{1/2}}{h^{3/4}}$	**(7.22a)**	Page 316
Centrifugal pump specific speed (in terms of head H):	$N_{s_{cu}} = \dfrac{\omega Q^{1/2}}{H^{3/4}}$	**(7.22b)**	Page 316

Case Study

T. Rex

Tyrannosaurus rex. (California Academy of Sciences)

Dimensional analysis, the main topic of this chapter, is used in many scientific pursuits. It has even been used by Professor Alexander McNeil, now at Heriot-Watt University in Scotland, to try to determine the speed at which dinosaurs such as *Tyrannosaurus rex* may have been able to run. The only data available on these creatures are in the fossil record—the most pertinent data being the dinosaurs' average leg length l and stride s. Could these data be used to extract the dinosaurs' speed? Comparing data on l and s and the speed V of quadrupeds (e.g., horses, dogs) and bipeds (e.g., humans) does not indicate a pattern, unless dimensional analysis is used to learn that all of the data should be plotted in the following way: Plot the dimensionless quantity V^2/gl (where V is the measured speed of the animal and g is the acceleration of gravity) against the dimensionless ratio s/l. When this is done, "magically" the data for most animals fall approximately on one curve! Hence, the running behavior of most animals can be obtained from the graph: In this case, the dinosaurs' value of s/l allows a corresponding value of V^2/gl to be interpolated from the curve, leading to an estimate for V of dinosaurs (because l and g are known). Based on this, in contrast to the *Jurassic Park* movies, it seems likely that humans could easily outrun *T. rex*!

References

1. Kline, S. J., *Similitude and Approximation Theory*. New York: McGraw-Hill, 1965.
2. Hansen, A. G., *Similarity Analysis of Boundary-Value Problems in Engineering*. Englewood Cliffs, NJ: Prentice-Hall, 1964.
3. Schlichting, H., *Boundary Layer Theory*, 7th ed. New York: McGraw-Hill, 1979.
4. Buckingham, E., "On Physically Similar Systems: Illustrations of the Use of Dimensional Equations," *Physical Review*, *4*, 4, 1914, pp. 345–376.
5. Todd, L. H., "Resistance and Propulsion," in *Principles of Naval Architecture*, J. P. Comstock, ed. New York: Society of Naval Architects and Marine Engineers, 1967.

6. "Aerodynamic Flow Visualization Techniques and Procedures." Warrendale, PA: Society of Automotive Engineers, SAE Information Report HS J1566, January 1986.
7. Merzkirch, W., *Flow Visualization*, 2nd ed. New York: Academic Press, 1987.
8. "SAE Wind Tunnel Test Procedure for Trucks and Buses," *Recommended Practice* SAE J1252, Warrendale, PA: Society of Automotive Engineers, 1981.
9. Sedov, L. I., *Similarity and Dimensional Methods in Mechanics*. New York: Academic Press, 1959.
10. Birkhoff, G., *Hydrodynamics—A Study in Logic, Fact, and Similitude*, 2nd ed. Princeton, NJ: Princeton University Press, 1960.
11. Ipsen, D. C., *Units, Dimensions, and Dimensionless Numbers*. New York: McGraw-Hill, 1960.
12. Yalin, M. S., *Theory of Hydraulic Models*. New York: Macmillan, 1971.
13. Pankhurst, R. C., and D. W. Holder, *Wind-Tunnel Technique*. London: Pitman, 1965.
14. Rae, W. H., and A. Pope, *Low-Speed Wind Tunnel Testing*, 2nd ed. New York: Wiley-Interscience, 1984.
15. Pope, A., and K. L. Goin, *High-Speed Wind Tunnel Testing*. New York: Krieger, 1978.
16. Waugh, J. G., and G. W. Stubstad, *Hydroballistics Modeling*. San Diego, CA: U.S. Naval Undersea Center, *ca*. 1965.
17. Baals, D. W., and W. R. Corliss, *Wind Tunnels of NASA*. Washington, D.C.: National Aeronautics and Space Administration, SP-440, 1981.
18. Vincent, M., "The Naval Ship Research and Development Center." Carderock, MD: Naval Ship Research and Development Center, Report 3039 (Revised), November 1971.
19. Smith, B. E., P. T. Zell, and P. M. Shinoda, "Comparison of Model- and Full-Scale Wind-Tunnel Performance," *Journal of Aircraft*, *27*, 3, March 1990, pp. 232–238.

Problems

Nondimensionalizing the Basic Differential Equations

Many of the Problems in this chapter involve obtaining the Π groups that characterize a problem. The *Excel* workbook used in Example 7.1 is useful for performing the computations involved. To avoid needless duplication, the computer symbol will only be used next to Problems when they have an *additional* benefit (e.g., for graphing).

7.1 The propagation speed of small-amplitude surface waves in a region of uniform depth is given by

$$c^2 = \left(\frac{\sigma}{\rho}\frac{2\pi}{\lambda} + \frac{g\lambda}{2\pi}\right)\tanh\frac{2\pi h}{\lambda}$$

where h is depth of the undisturbed liquid and λ is wavelength. Using L as a characteristic length and V_0 as a characteristic velocity, obtain the dimensionless groups that characterize the equation.

7.2 The equation describing small-amplitude vibration of a beam is

$$\rho A\frac{\partial^2 y}{\partial t^2} + EI\frac{\partial^4 y}{\partial x^4} = 0$$

where y is the beam deflection at location x and time t, ρ and E are the density and modulus of elasticity of the beam material, respectively, and A and I are the beam cross-section area and second moment of area, respectively. Use the beam length L, and frequency of vibration ω, to nondimensionalize this equation. Obtain the dimensionless groups that characterize the equation.

7.3 The slope of the free surface of a steady wave in one-dimensional flow in a shallow liquid layer is described by the equation

$$\frac{\partial h}{\partial x} = -\frac{u}{g}\frac{\partial u}{\partial x}$$

Use a length scale, L, and a velocity scale, V_0, to nondimensionalize this equation. Obtain the dimensionless groups that characterize this flow.

7.4 One-dimensional unsteady flow in a thin liquid layer is described by the equation

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} = -g\frac{\partial h}{\partial x}$$

Use a length scale, L, and a velocity scale, V_0, to nondimensionalize this equation. Obtain the dimensionless groups that characterize this flow.

7.5 A two-dimensional steady flow in a viscous liquid is described by the equation:

$$u\frac{\partial u}{\partial x} = -g\frac{\partial h}{\partial x} + \frac{\mu}{\rho}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)$$

Use a length scale, L, and a velocity scale, V_0, to nondimensionalize this equation. Obtain the dimensionless groups that characterize this flow.

7.6 In atmospheric studies the motion of the earth's atmosphere can sometimes be modeled with the equation

$$\frac{D\vec{V}}{Dt} + 2\vec{\Omega}\times\vec{V} = -\frac{1}{\rho}\nabla p$$

where $\vec{V}$ is the large-scale velocity of the atmosphere across the Earth's surface, ∇p is the climatic pressure gradient, and $\vec{\Omega}$ is the Earth's angular velocity. What is the meaning of the term $\vec{\Omega}\times\vec{V}$? Use the pressure difference, Δp, and typical length scale, L (which could, for example, be the magnitude of, and distance between, an atmospheric high and low, respectively), to nondimensionalize this equation. Obtain the dimensionless groups that characterize this flow.

7.7 By using order of magnitude analysis, the continuity and Navier–Stokes equations can be simplified to the Prandtl

boundary-layer equations. For steady, incompressible, and two-dimensional flow, neglecting gravity, the result is

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\frac{\partial^2 u}{\partial y^2}$$

Use L and V_0 as characteristic length and velocity, respectively. Nondimensionalize these equations and identify the similarity parameters that result.

7.8 An unsteady, two-dimensional, compressible, inviscid flow can be described by the equation

$$\frac{\partial^2 \psi}{\partial t^2} + \frac{\partial}{\partial t}(u^2 + v^2) + (u^2 - c^2)\frac{\partial^2 \psi}{\partial x^2} + (v^2 - c^2)\frac{\partial^2 \psi}{\partial y^2} + 2uv\frac{\partial^2 \psi}{\partial x \partial y} = 0$$

where ψ is the stream function, u and v are the x and y components of velocity, respectively, c is the local speed of sound, and t is the time. Using L as a characteristic length and c_0 (the speed of sound at the stagnation point) to nondimensionalize this equation, obtain the dimensionless groups that characterize the equation.

7.9 The equation describing motion of fluid in a pipe due to an applied pressure gradient, when the flow starts from rest, is

$$\frac{\partial u}{\partial t} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\left(\frac{\partial^2 u}{\partial r^2} + \frac{1}{r}\frac{\partial u}{\partial r}\right)$$

Use the average velocity $\overline{V}$, pressure drop Δp, pipe length L, and diameter D to nondimensionalize this equation. Obtain the dimensionless groups that characterize this flow.

Determining the Π Groups

7.10 Experiments show that the pressure drop for flow through an orifice plate of diameter d mounted in a length of pipe of diameter D may be expressed as $\Delta p = p_1 - p_2 = f(\rho, \mu, \overline{V}, d, D)$. You are asked to organize some experimental data. Obtain the resulting dimensionless parameters.

7.11 At relatively high speeds the drag on an object is independent of fluid viscosity. Thus the aerodynamic drag force, F, on an automobile, is a function only of speed, V, air density ρ, and vehicle size, characterized by its frontal area A. Use dimensional analysis to determine how the drag force F depends on the speed V.

7.12 At very low speeds, the drag on an object is independent of fluid density. Thus the drag force, F, on a small sphere is a function only of speed, V, fluid viscosity, μ, and sphere diameter, D. Use dimensional analysis to determine how the drag force F depends on the speed V.

7.13 The drag force on the International Space Station depends on the mean free path of the molecules λ (a length), the density ρ, a characteristic length L, and the mean speed of the air molecules c. Find a nondimensional form of this functional relationship.

7.14 We saw in Chapter 3 that the buoyant force, F_B, on a body submerged in a fluid is directly proportional to the specific weight of the fluid, γ. Demonstrate this using dimensional analysis, by starting with the buoyant force as a function of the volume of the body and the specific weight of the fluid.

7.15 When an object travels at supersonic speeds, the aerodynamic drag force F acting on the object is a function of the velocity V, air density ρ, object size (characterized by some reference area A), and the speed of sound c (note that all of the variables except c were considered when traveling at subsonic speeds as in Problem 7.11). Develop a functional relationship between a set of dimensionless variables to describe this problem.

7.16 The speed, V, of a free-surface wave in shallow liquid is a function of depth, D, density, ρ, gravity, g, and surface tension, σ. Use dimensional analysis to find the functional dependence of V on the other variables. Express V in the simplest form possible.

7.17 The wall shear stress, τ_w, in a boundary layer depends on distance from the leading edge of the body, x, the density, ρ, and viscosity, μ, of the fluid, and the freestream speed of the flow, U. Obtain the dimensionless groups and express the functional relationship among them.

7.18 The boundary-layer thickness, δ, on a smooth flat plate in an incompressible flow without pressure gradients depends on the freestream speed, U, the fluid density, ρ, the fluid viscosity, μ, and the distance from the leading edge of the plate, x. Express these variables in dimensionless form.

7.19 If an object is light enough it can be supported on the surface of a fluid by surface tension. Tests are to be done to investigate this phenomenon. The weight, W, supportable in this way depends on the object's perimeter, p, and the fluid's density, ρ, surface tension σ, and gravity, g. Determine the dimensionless parameters that characterize this problem.

7.20 The speed, V, of a free-surface gravity wave in deep water is a function of wavelength, λ, depth, D, density, ρ, and acceleration of gravity, g. Use dimensional analysis to find the functional dependence of V on the other variables. Express V in the simplest form possible.

7.21 The mean velocity, $\overline{u}$, for turbulent flow in a pipe or a boundary layer may be correlated using the wall shear stress, τ_w, distance from the wall, y, and the fluid properties, ρ and μ. Use dimensional analysis to find one dimensionless parameter containing $\overline{u}$ and one containing y that are suitable for organizing experimental data. Show that the result may be written

$$\frac{\overline{u}}{u_*} = f\left(\frac{yu_*}{\nu}\right)$$

where $u_* = (\tau_w/\rho)^{1/2}$ is the *friction velocity*.

7.22 The energy released during an explosion, E, is a function of the time after detonation t, the blast radius R at time t, and the ambient air pressure p, and density ρ. Determine, by dimensional analysis, the general form of the expression for E in terms of the other variables.

7.23 Capillary waves are formed on a liquid free surface as a result of surface tension. They have short wavelengths. The speed of a capillary wave depends on surface tension, σ, wavelength, λ, and liquid density, ρ. Use dimensional analysis to express wave speed as a function of these variables.

7.24 Measurements of the liquid height upstream from an obstruction placed in an open-channel flow can be used to determine volume flow rate. (Such obstructions, designed and calibrated to measure rate of open-channel flow, are called *weirs*.) Assume the volume flow rate, Q, over a weir is a function of upstream height, h, gravity, g, and channel width, b. Use dimensional analysis to find the functional dependence of Q on the other variables.

7.25 The torque, T, of a handheld automobile buffer is a function of rotational speed, ω, applied normal force, F, automobile surface roughness, e, buffing paste viscosity, μ, and surface tension, σ. Determine the dimensionless parameters that characterize this problem.

7.26 The power, $\mathcal{P}$, used by a vacuum cleaner is to be correlated with the amount of suction provided (indicated by the pressure drop, Δp, below the ambient room pressure). It also depends on impeller diameter, D, and width, d, motor speed, ω, air density, ρ, and cleaner inlet and exit widths, d_i and d_o, respectively. Determine the dimensionless parameters that characterize this problem.

7.27 The load-carrying capacity, W, of a journal bearing is known to depend on its diameter, D, length, l, and clearance, c, in addition to its angular speed, ω, and lubricant viscosity, μ. Determine the dimensionless parameters that characterize this problem.

7.28 The time, t, for oil to drain out of a viscosity calibration container depends on the fluid viscosity, μ, and density, ρ, the orifice diameter, d, and gravity, g. Use dimensional analysis to find the functional dependence of t on the other variables. Express t in the simplest possible form.

7.29 The power per unit cross-sectional area, E, transmitted by a sound wave is a function of wave speed, V, medium density, ρ, wave amplitude, r, and wave frequency, n. Determine, by dimensional analysis, the general form of the expression for E in terms of the other variables.

7.30 You are asked to find a set of dimensionless parameters to organize data from a laboratory experiment, in which a tank is drained through an orifice from initial liquid level h_0. The time, τ, to drain the tank depends on tank diameter, D, orifice diameter, d, acceleration of gravity, g, liquid density, ρ, and liquid viscosity, μ. How many dimensionless parameters will result? How many repeating variables must be selected to determine the dimensionless parameters? Obtain the Π parameter that contains the viscosity.

7.31 A continuous belt moving vertically through a bath of viscous liquid drags a layer of liquid, of thickness h, along with it. The volume flow rate of liquid, Q, is assumed to depend on μ, ρ, g, h, and V, where V is the belt speed. Apply dimensional analysis to predict the form of dependence of Q on the other variables.

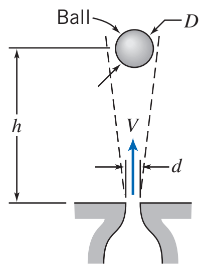
7.32 The power, $\mathcal{P}$, required to drive a fan is believed to depend on fluid density, ρ, volume flow rate, Q, impeller diameter, D, and angular velocity, ω. Use dimensional analysis to determine the dependence of $\mathcal{P}$ on the other variables.

7.33 In a fluid mechanics laboratory experiment a tank of water, with diameter D, is drained from initial level h_0. The smoothly rounded drain hole has diameter d. Assume the mass flow rate from the tank is a function of h, D, d, g, ρ, and μ, where g is the acceleration of gravity and ρ and μ are fluid properties. Measured data are to be correlated in dimensionless form. Determine the number of dimensionless parameters that will result. Specify the number of repeating parameters that must be selected to determine the dimensionless parameters. Obtain the Π parameter that contains the viscosity.

7.34 Cylindrical water tanks are frequently found on the tops of tall buildings. When a tank is filled with water, the bottom of the tank typically deflects under the weight of the water inside. The deflection δ is a function of the tank diameter D, the height of water h, the thickness of the tank bottom d, the specific weight of the water γ, and the modulus of elasticity of the tank material E. Determine the functional relationship among these parameters using dimensionless groups.

7.35 Small droplets of liquid are formed when a liquid jet breaks up in spray and fuel injection processes. The resulting droplet diameter, d, is thought to depend on liquid density, viscosity, and surface tension, as well as jet speed, V, and diameter, D. How many dimensionless ratios are required to characterize this process? Determine these ratios.

7.36 The sketch shows an air jet discharging vertically. Experiments show that a ball placed in the jet is suspended in a stable position. The equilibrium height of the ball in the jet is found to depend on D, d, V, ρ, μ, and W, where W is the weight of the ball. Dimensional analysis is suggested to correlate experimental data. Find the Π parameters that characterize this phenomenon.



P7.36

7.37 The diameter, d, of the dots made by an ink jet printer depends on the ink viscosity, μ, density, ρ, and surface tension, σ, the nozzle diameter, D, the distance, L, of the nozzle from the paper surface, and the ink jet velocity, V. Use dimensional analysis to find the Π parameters that characterize the ink jet's behavior.

7.38 The diameter, d, of bubbles produced by a bubble-making toy depends on the soapy water viscosity, μ, density, ρ, and surface tension, σ, the ring diameter, D, and the pressure differential, Δp, generating the bubbles. Use dimensional analysis to find the Π parameters that characterize this phenomenon.

7.39 The terminal speed V of shipping boxes sliding down an incline on a layer of air (injected through numerous pinholes in the incline surface) depends on the box mass, m, and base area, A, gravity, g, the incline angle, θ, the air viscosity, μ, and the air layer thickness, δ. Use dimensional analysis to find the Π parameters that characterize this phenomenon.

7.40 The length of the wake w behind an airfoil is a function of the flow speed V, chord length L, thickness t, and fluid density ρ and viscosity μ. Find the dimensionless parameters that characterize this phenomenon.

7.41 A washing machine agitator is to be designed. The power, $\mathcal{P}$, required for the agitator is to be correlated with the amount of water used (indicated by the depth, H, of the water). It also depends on the agitator diameter, D, height, h, maximum angular velocity, $\omega_{\max}$, and frequency of oscillations, f, and water density, ρ, and viscosity, μ. Determine the dimensionless parameters that characterize this problem.

7.42 Choked-flow nozzles are often used to meter the flow of gases through piping systems. The mass flow rate of gas is thought to depend on nozzle area A, pressure p, and temperature T upstream of the meter, and the gas constant R. Determine how many independent Π parameters can be formed for this problem. State the functional relationship for the mass flow rate in terms of the dimensionless parameters.

7.43 The time, t, for a flywheel, with moment of inertia, I, to reach angular velocity, ω, from rest, depends on the applied torque, T, and the following flywheel bearing properties: the oil viscosity, μ, gap, δ, diameter, D, and length, L. Use dimensional analysis to find the Π parameters that characterize this phenomenon.

7.44 A large tank of liquid under pressure is drained through a smoothly contoured nozzle of area A. The mass flow rate is thought to depend on nozzle area, A, liquid density, ρ, difference in height between the liquid surface and nozzle, h, tank gage pressure, Δp, and gravitational acceleration, g. Determine how many independent Π parameters can be formed for this problem. Find the dimensionless parameters. State the functional relationship for the mass flow rate in terms of the dimensionless parameters.

7.45 Spin plays an important role in the flight trajectory of golf, Ping-Pong, and tennis balls. Therefore, it is important to know the rate at which spin decreases for a ball in flight. The aerodynamic torque, T, acting on a ball in flight, is thought to depend on flight speed, V, air density, ρ, air viscosity, μ, ball diameter, D, spin rate (angular speed), ω, and diameter of the dimples on the ball, d. Determine the dimensionless parameters that result.

7.46 The ventilation in the clubhouse on a cruise ship is insufficient to clear cigarette smoke (the ship is not yet completely smoke-free). Tests are to be done to see if a larger extractor fan will work. The concentration of smoke, c (particles per cubic meter) depends on the number of smokers, N, the pressure drop produced by the fan, Δp, the fan diameter, D, motor speed, ω, the particle and air densities, ρ_p, and ρ, respectively, gravity, g, and air viscosity, μ. Determine the dimensionless parameters that characterize this problem.

7.47 The mass burning rate of flammable gas $\dot{m}$ is a function of the thickness of the flame δ, the gas density ρ, the thermal diffusivity α, and the mass diffusivity D. Using dimensional analysis, determine the functional form of this dependence in terms of dimensionless parameters. Note that α and D have the dimensions L^2/t.

7.48 The power loss, $\mathcal{P}$, in a journal bearing depends on length, l, diameter, D, and clearance, c, of the bearing, in addition to its angular speed, ω. The lubricant viscosity and mean pressure are also important. Obtain the dimensionless parameters that characterize this problem. Determine the functional form of the dependence of $\mathcal{P}$ on these parameters.

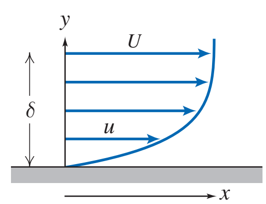
7.49 In a fan-assisted convection oven, the heat transfer rate to a roast, $\dot{Q}$ (energy per unit time), is thought to depend on the specific heat of air, c_p, temperature difference, Θ, a length scale, L, air density, ρ, air viscosity, μ, and air speed, V. How many basic dimensions are included in these variables? Determine the number of Π parameters needed to characterize the oven. Evaluate the Π parameters.

7.50 The thrust of a marine propeller is to be measured during "open-water" tests at a variety of angular speeds and forward speeds ("speeds of advance"). The thrust, F_T, is thought to depend on water density, ρ, propeller diameter, D, speed of advance, V, acceleration of gravity, g, angular speed, ω, pressure in the liquid, p, and liquid viscosity, μ. Develop a set of dimensionless parameters to characterize the performance of the propeller. (One of the resulting parameters, gD/V^2, is known as the *Froude speed of advance*.)

7.51 The rate dT/dt at which the temperature T at the center of a rice kernel falls during a food technology process is critical—too high a value leads to cracking of the kernel, and too low a value makes the process slow and costly. The rate depends on the rice specific heat, c, thermal conductivity, k, and size, L, as well as the cooling air specific heat, c_p, density, ρ, viscosity, μ, and speed, V. How many basic dimensions are included in these variables? Determine the Π parameters for this problem.

7.52 The power, $\mathcal{P}$, required to drive a propeller is known to depend on the following variables: freestream speed, V, propeller diameter, D, angular speed, ω, fluid viscosity, μ, fluid density, ρ, and speed of sound in the fluid, c. How many dimensionless groups are required to characterize this situation? Obtain these dimensionless groups.

7.53 The fluid velocity u at any point in a boundary layer depends on the distance y of the point above the surface, the free-stream velocity U and free-stream velocity gradient dU/dx, the fluid kinematic viscosity ν, and the boundary layer thickness δ. How many dimensionless groups are required to describe this problem? Find: (a) two Π groups by inspection, (b) one Π that is a standard fluid mechanics group, and (c) any remaining Π groups using the Buckingham Pi theorem.



P7.53

7.54 When a valve is closed suddenly in a pipe with flowing water, a water hammer pressure wave is set up. The very high pressures generated by such waves can damage the pipe. The maximum pressure, $p_{\max}$, generated by water hammer is a function of liquid density, ρ, initial flow speed, U_0, and liquid bulk modulus, E_v. How many dimensionless groups

are needed to characterize water hammer? Determine the functional relationship among the variables in terms of the necessary Π groups.

Flow Similarity and Model Studies

7.55 The designers of a large tethered pollution-sampling balloon wish to know what the drag will be on the balloon for the maximum anticipated wind speed of 5 m/s (the air is assumed to be at 20°C). A $\frac{1}{20}$-scale model is built for testing in water at 20°C. What water speed is required to model the prototype? At this speed the model drag is measured to be 2 kN. What will be the corresponding drag on the prototype?

7.56 An airship is to operate at 20 m/s in air at standard conditions. A model is constructed to $\frac{1}{20}$-scale and tested in a wind tunnel at the same air temperature to determine drag. What criterion should be considered to obtain dynamic similarity? If the model is tested at 75 m/s, what pressure should be used in the wind tunnel? If the model drag force is 250 N, what will be the drag of the prototype?

7.57 To match the Reynolds number in an air flow and a water flow using the same size model, which flow will require the higher flow speed? How much higher must it be?

7.58 An ocean-going vessel is to be powered by a rotating circular cylinder. Model tests are planned to estimate the power required to rotate the prototype cylinder. A dimensional analysis is needed to scale the power requirements from model test results to the prototype. List the parameters that should be included in the dimensional analysis. Perform a dimensional analysis to identify the important dimensionless groups.

7.59 Measurements of drag force are made on a model automobile in a towing tank filled with fresh water. The model length scale is $\frac{1}{5}$ that of the prototype. State the conditions required to ensure dynamic similarity between the model and prototype. Determine the fraction of the prototype speed in air at which the model test should be made in water to ensure dynamically similar conditions. Measurements made at various speeds show that the dimensionless force ratio becomes constant at model test speeds above $V_m = 4$ m/s. The drag force measured during a test at this speed is $F_{D_m} = 182$ N. Calculate the drag force expected on the prototype vehicle operating at 90 km/hr in air.

7.60 On a cruise ship, passengers complain about the noise emanating from the ship's propellers (probably due to turbulent flow effects between the propeller and the ship). You have been hired to find out the source of this noise. You will study the flow pattern around the propellers and have decided to use a 1:9-scale water tank. If the ship's propellers rotate at 100 rpm, estimate the model propeller rotation speed if (a) the Froude number or (b) the Reynolds number is the governing dimensionless group. Which is most likely to lead to the best modeling?

7.61 A $\frac{1}{5}$-scale model of a torpedo is tested in a wind tunnel to determine the drag force. The prototype operates in water, has 533 mm diameter, and is 6.7 m long. The desired operating speed of the prototype is 28 m/s. To avoid compressibility effects in the wind tunnel, the maximum speed is limited to 110 m/s. However, the pressure in the wind tunnel can be varied while holding the temperature constant at 20°C. At what minimum pressure should the wind tunnel be operated to achieve a dynamically similar test? At dynamically similar test conditions, the drag force on the model is measured as 618 N. Evaluate the drag force expected on the full-scale torpedo.

7.62 The drag of an airfoil at zero angle of attack is a function of density, viscosity, and velocity, in addition to a length parameter. A 1:5-scale model of an airfoil was tested in a wind tunnel at a speed of 130 ft/s, temperature of 59°F, and 5 atmospheres absolute pressure. The prototype airfoil has a chord length of 6 ft and is to be flown in air at standard conditions. Determine the Reynolds number at which the wind tunnel model was tested and the corresponding prototype speed at the same Reynolds number.

7.63 Consider a smooth sphere, of diameter D, immersed in a fluid moving with speed V. The drag force on a 10-ft-diameter weather balloon in air moving at 5 ft/s is to be calculated from test data. The test is to be performed in water using a 2-in.-diameter model. Under dynamically similar conditions, the model drag force is measured as 0.85 lbf. Evaluate the model test speed and the drag force expected on the full-scale balloon.

7.64 An airplane wing, with chord length of 1.5 m and span of 9 m, is designed to move through standard air at a speed of 7.5 m/s. A $\frac{1}{10}$-scale model of this wing is to be tested in a water tunnel. What speed is necessary in the water tunnel to achieve dynamic similarity? What will be the ratio of forces measured in the model flow to those on the prototype wing?

7.65 The fluid dynamic characteristics of a golf ball are to be tested using a model in a wind tunnel. Dependent parameters are the drag force, F_D, and lift force, F_L, on the ball. The independent parameters should include angular speed, ω, and dimple depth, d. Determine suitable dimensionless parameters and express the functional dependence among them. A golf pro can hit a ball at V = 75 m/s and ω = 8100 rpm. To model these conditions in a wind tunnel with a maximum speed of 25 m/s, what diameter model should be used? How fast must the model rotate? (The diameter of a U.S. golf ball is 4.27 cm.)

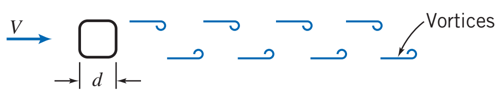
7.66 A water pump with impeller diameter 24 in. is to be designed to move 15 ft^3/s when running at 750 rpm. Testing is performed on a 1:4 scale model running at 2400 rpm using air (68°F) as the fluid. For similar conditions (neglecting Reynolds number effects), what will be the model flow rate? If the model draws 0.1 hp, what will be the power requirement of the prototype?

7.67 A model test is performed to determine the flight characteristics of a Frisbee. Dependent parameters are drag force, F_D, and lift force, F_L. The independent parameters should include angular speed, ω, and roughness height, h. Determine suitable dimensionless parameters, and express the functional dependence among them. The test (using air) on a 1:7-scale model Frisbee is to be geometrically, kinematically, and dynamically similar to the prototype. The wind tunnel test conditions are V_m = 140 ft/s and ω_m = 5000 rpm. What are the corresponding values of V_p and ω_p?

7.68 A model hydrofoil is to be tested at 1:20 scale. The test speed is chosen to duplicate the Froude number corresponding to the 60-knot prototype speed. To model cavitation correctly, the cavitation number also must be duplicated. At what ambient pressure must the test be run? Water in the model test basin can be heated to 130°F, compared to 45°F for the prototype.

7.69 SAE 10W oil at 77°F flowing in a 1-in.-diameter horizontal pipe, at an average speed of 3 ft/s, produces a pressure drop of 7 psi (gage) over a 500-ft length. Water at 60°F flows through the same pipe under dynamically similar conditions. Using the results of Example 7.2, calculate the average speed of the water flow and the corresponding pressure drop.

7.70 In some speed ranges, vortices are shed from the rear of bluff cylinders placed across a flow. The vortices alternately leave the top and bottom of the cylinder, as shown, causing an alternating force normal to the freestream velocity. The vortex shedding frequency, f, is thought to depend on ρ, d, V, and μ. Use dimensional analysis to develop a functional relationship for f. Vortex shedding occurs in standard air on two cylinders with a diameter ratio of 2. Determine the velocity ratio for dynamic similarity, and the ratio of vortex shedding frequencies.



P7.70

7.71 A $\frac{1}{8}$-scale model of a tractor-trailer rig is tested in a pressurized wind tunnel. The rig width, height, and length are $W = 0.305$ m, $H = 0.476$ m, and $L = 2.48$ m, respectively. At wind speed $V = 75.0$ m/s, the model drag force is $F_D = 128$ N. (Air density in the tunnel is $\rho = 3.23$ kg/m^3.) Calculate the aerodynamic drag coefficient for the model. Compare the Reynolds numbers for the model test and for the prototype vehicle at 55 mph. Calculate the aerodynamic drag force on the prototype vehicle at a road speed of 55 mph into a headwind of 10 mph.

7.72 On a cruise ship, passengers complain about the amount of smoke that becomes entrained behind the cylindrical smoke stack. You have been hired to study the flow pattern around the stack, and have decided to use a 1:15 scale model of the 15-ft smoke stack. What range of wind tunnel speeds could you use if the ship speed for which the problem occurs is 12 to 24 knots?

7.73 The aerodynamic behavior of a flying insect is to be investigated in a wind tunnel using a 1:8-scale model. If the insect flaps its wings 60 times per second when flying at 1.5 m/s, determine the wind tunnel air speed and wing oscillation required for dynamic similarity. Do you expect that this would be a successful or practical model for generating an easily measurable wing lift? If not, can you suggest a different fluid (e.g., water, or air at a different pressure or temperature) that would produce a better modeling?

7.74 A model test of a tractor-trailer rig is performed in a wind tunnel. The drag force, F_D, is found to depend on frontal area A, wind speed V, air density ρ, and air viscosity μ. The model scale is 1:4; frontal area of the model is 7 ft^2. Obtain a set of dimensionless parameters suitable to characterize the model test results. State the conditions required to obtain dynamic similarity between model and prototype flows. When tested at wind speed V = 300 ft/s in standard air, the measured drag force on the model was F_D = 550 lbf. Assuming dynamic similarity, estimate the aerodynamic drag force on the full-scale vehicle at V = 75 ft/s. Calculate the power needed to overcome this drag force if there is no wind.

7.75 Tests are performed on a 1:10-scale boat model. What must be the kinematic viscosity of the model fluid if friction and wave drag phenomena are to be correctly modeled? The full-size boat will be used in a freshwater lake where the average water temperature is 50°F.

7.76 Your favorite professor likes mountain climbing, so there is always a possibility that the professor may fall into a crevasse in some glacier. If that happened today, and the professor was trapped in a slowly moving glacier, you are curious to know whether the professor would reappear at the downstream drop-off of the glacier during this academic year. Assuming ice is a Newtonian fluid with the density of glycerine but a million times as viscous, you decide to build a glycerin model and use dimensional analysis and similarity to estimate when the professor would reappear. Assume the real glacier is 15 m deep and is on a slope that falls 1.5 m in a horizontal distance of 1850 m. Develop the dimensionless parameters and conditions expected to govern dynamic similarity in this problem. If the model professor reappears in the laboratory after 9.6 hours, when should you return to the end of the real glacier to provide help to your favorite professor?

7.77 An automobile is to travel through standard air at 60 mph. To determine the pressure distribution, a $\frac{1}{5}$-scale model is to be tested in water. What factors must be considered to ensure kinematic similarity in the tests? Determine the water speed that should be used. What is the corresponding ratio of drag force between prototype and model flows? The lowest pressure coefficient is $C_p = -1.4$ at the location of the minimum static pressure on the surface. Estimate the minimum tunnel pressure required to avoid cavitation, if the onset of cavitation occurs at a cavitation number of 0.5.

7.78 A 1:50-scale model of a submarine is to be tested in a towing tank under two conditions: motion at the free surface and motion far below the surface. The tests are performed in freshwater. On the surface, the submarine cruises at 24 knots. At what speed should the model be towed to ensure dynamic similarity? Far below the surface, the sub cruises at 0.35 knot. At what speed should the model be towed to ensure dynamic similarity? What must the drag of the model be multiplied by under each condition to give the drag of the full-scale submarine?

7.79 A wind tunnel is being used to study the aerodynamics of a full-scale model rocket that is 12 in. long. Scaling for drag calculations are based on the Reynolds number. The rocket has an expected maximum velocity of 120 mph. What is the Reynolds number at this speed? Assume ambient air is at 68°F. The wind tunnel is capable of speeds up to 100 mph; so an attempt is made to improve this top speed by varying

the air temperature. Calculate the equivalent speed for the wind tunnel using air at 40°F and 150°F. Would replacing air with carbon dioxide provide higher equivalent speeds?

7.80 Consider water flow around a circular cylinder, of diameter D and length l. In addition to geometry, the drag force is known to depend on liquid speed, V, density, ρ, and viscosity, μ. Express drag force, F_D, in dimensionless form as a function of all relevant variables. The static pressure distribution on a circular cylinder, measured in the laboratory, can be expressed in terms of the dimensionless pressure coefficient; the lowest pressure coefficient is $C_p = -2.4$ at the location of the minimum static pressure on the cylinder surface. Estimate the maximum speed at which a cylinder could be towed in water at atmospheric pressure, without causing cavitation, if the onset of cavitation occurs at a cavitation number of 0.5.

7.81 A circular container, partially filled with water, is rotated about its axis at constant angular speed, ω. At any time, τ, from the start of rotation, the speed, V_θ, at distance r from the axis of rotation, was found to be a function of τ, ω, and the properties of the liquid. Write the dimensionless parameters that characterize this problem. If, in another experiment, honey is rotated in the same cylinder at the same angular speed, determine from your dimensionless parameters whether honey will attain steady motion as quickly as water. Explain why the Reynolds number would not be an important dimensionless parameter in scaling the steady-state motion of liquid in the container.

7.82 A $\frac{1}{10}$-scale model of a tractor-trailer rig is tested in a wind tunnel. The model frontal area is $A_m = 0.1\ \text{m}^2$. When tested at $V_m = 75$ m/s in standard air, the measured drag force is $F_D = 350$ N. Evaluate the drag coefficient for the model conditions given. Assuming that the drag coefficient is the same for model and prototype, calculate the drag force on a prototype rig at a highway speed of 90 km/hr. Determine the air speed at which a model should be tested to ensure dynamically similar results if the prototype speed is 90 km/hr. Is this air speed practical? Why or why not?

7.83 It is recommended in [8] that the frontal area of a model be less than 5 percent of the wind tunnel test section area and $Re = Vw/\nu > 2 \times 10^6$, where w is the model width. Further, the model height must be less than 30 percent of the test section height, and the maximum projected width of the model at maximum yaw (20°) must be less than 30 percent of the test section width. The maximum air speed should be less than 300 ft/s to avoid compressibility effects. A model of a tractor-trailer rig is to be tested in a wind tunnel that has a test section 1.5 ft high and 2 ft wide. The height, width, and length of the full-scale rig are 13 ft 6 in., 8 ft, and 65 ft, respectively. Evaluate the scale ratio of the largest model that meets the recommended criteria. Assess whether an adequate Reynolds number can be achieved in this test facility.

7.84 The power, $\mathcal{P}$, required to drive a fan is assumed to depend on fluid density ρ, volume flow rate Q, impeller diameter D, and angular speed ω. If a fan with $D_1 = 8$ in. delivers $Q_1 = 15\ \text{ft}^3/\text{s}$ of air at $\omega_1 = 2500$ rpm, what size diameter fan could be expected to deliver $Q_2 = 88\ \text{ft}^3/\text{s}$ of air at $\omega_2 = 1800$ rpm, provided they were geometrically and dynamically similar?

7.85 Over a certain range of air speeds, V, the lift, F_L, produced by a model of a complete aircraft in a wind tunnel depends on the air speed, air density, ρ, and a characteristic length (the wing base chord length, $c = 150$ mm). The following experimental data is obtained for air at standard atmospheric conditions:

V **(m/s)**	10	15	20	25	30	35	40	45	50
F_L **(N)**	2.2	4.8	8.7	13.3	19.6	26.5	34.5	43.8	54

Plot the lift versus speed curve. By using *Excel* to perform a trendline analysis on this curve, generate and plot data for the lift produced by the prototype, which has a wing base chord length of 5 m, over a speed range of 75 m/s to 250 m/s.

7.86 The pressure rise, Δp, of a liquid flowing steadily through a centrifugal pump depends on pump diameter D, angular speed of the rotor ω, volume flow rate Q, and density ρ. The table gives data for the prototype and for a geometrically similar model pump. For conditions corresponding to dynamic similarity between the model and prototype pumps, calculate the missing values in the table.

Variable	**Prototype**	**Model**
Δp	52.5 kPa	
Q		$0.0928\ \text{m}^3/\text{min}$
ρ	$800\ \text{kg/m}^3$	$999\ \text{kg/m}^3$
ω	183 rad/s	367 rad/s
D	150 mm	50 mm

7.87 Tests are performed on a 3-ft-long ship model in a water tank. Results obtained (after doing some data analysis) are as follows:

V **(ft/s)**	10	20	30	40	50	60	70
D_{Wave} **(lbf)**	0	0.028	0.112	0.337	0.674	0.899	1.237
D_{Friction} **(lbf)**	0.022	0.079	0.169	0.281	0.45	0.618	0.731

The assumption is that wave drag is done using the Froude number and friction drag by the Reynolds number. The full-size ship will be 150 ft long when built. Estimate the total drag when it is cruising at 15 knots and at 20 knots in a freshwater lake.

7.88 A centrifugal water pump running at speed $\omega = 800$ rpm has the following data for flow rate, Q, and pressure head, Δp.

Q **(ft³/min)**	0	50	75	100	120	140	150	165
Δp **(psf)**	7.54	7.29	6.85	6.12	4.80	3.03	2.38	1.23

The pressure head is a function of the flow rate, speed, impeller diameter D, and water density ρ. Plot the pressure head versus flow rate curve. Find the two Π parameters for this problem, and, from the above data, plot one against the other. By using *Excel* to perform a trendline analysis on this latter curve, generate and plot data for pressure head versus flow rate for impeller speeds of 600 rpm and 1200 rpm.

7.89 An axial-flow pump is required to deliver $0.75\ \text{m}^3/\text{s}$ of water at a head of 15 J/kg. The diameter of the rotor is 0.25 m, and it is to be driven at 500 rpm. The prototype is to be modeled

on a small test apparatus having a 2.25 kW, 1000 rpm power supply. For similar performance between the prototype and the model, calculate the head, volume flow rate, and diameter of the model.

7.90 A model propeller 1 m in diameter is tested in a wind tunnel. Air approaches the propeller at 50 m/s when it rotates at 1800 rpm. The thrust and torque measured under these conditions are 100 N and 10 N·m, respectively. A prototype 8 times as large as the model is to be built. At a dynamically similar operating point, the approach air speed is to be 130 m/s. Calculate the speed, thrust, and torque of the prototype propeller under these conditions, neglecting the effect of viscosity but including density.

7.91 Consider again Problem 7.51. Experience shows that for ship-size propellers, viscous effects on scaling are small. Also, when cavitation is not present, the nondimensional parameter containing pressure can be ignored. Assume that torque, T, and power, $\mathscr{P}$, depend on the same parameters as thrust. For conditions under which effects of μ and p can be neglected, derive scaling "laws" for propellers, similar to the pump "laws" of Section 7.6, that relate thrust, torque, and power to the angular speed and diameter of the propeller.

7.92 Water drops are produced by a mechanism that it is believed follows the pattern $d_p = D(\text{We})^{-3/5}$. In this formula, d_p is the drop size, D is proportional to a length scale, and We is the Weber number. In scaling up, if the large-scale characteristic length scale was increased by a factor of 20 and the large-scale velocity decreased by a factor of 5, how would the small- and large-scale drops differ from each other for the same material, for example, water?

7.93 Closed-circuit wind tunnels can produce higher speeds than open-circuit tunnels with the same power input because energy is recovered in the diffuser downstream from the test section. The *kinetic energy ratio* is a figure of merit defined as the ratio of the kinetic energy flux in the test section to the drive power. Estimate the kinetic energy ratio for the 40 ft × 80 ft wind tunnel at NASA-Ames described on page 318.

7.94 A 1:16 model of a 60-ft-long truck is tested in a wind tunnel at a speed of 250 ft/s, where the axial static pressure gradient is -0.07 lbf/ft^2 per foot. The frontal area of the prototype is 110 ft^2. Estimate the horizontal buoyancy correction for this situation. Express the correction as a fraction of the measured C_D, of $C_D = 0.85$.

7.95 Frequently one observes a flag on a pole flapping in the wind. Explain why this occurs.

7.96 A 1:16 model of a bus is tested in a wind tunnel in standard air. The model is 152 mm wide, 200 mm high, and 762 mm long. The measured drag force at 26.5 m/s wind speed is 6.09 N. The longitudinal pressure gradient in the wind tunnel test section is -11.8 N/m^2/m. Estimate the correction that should be made to the measured drag force to correct for horizontal buoyancy caused by the pressure gradient in the test section. Calculate the drag coefficient for the model. Evaluate the aerodynamic drag force on the prototype at 100 km/hr on a calm day.

7.97 Explore the variation in wave propagation speed given by the equation of Problem 7.1 for a free-surface flow of water. Find the operating depth to minimize the speed of capillary waves (waves with small wavelength, also called ripples). First assume wavelength is much smaller than water depth. Then explore the effect of depth. What depth do you recommend for a water table used to visualize compressible-flow wave phenomena? What is the effect of reducing surface tension by adding a surfactant?

8

Internal Incompressible Viscous Flow

8.12 Traversing Methods

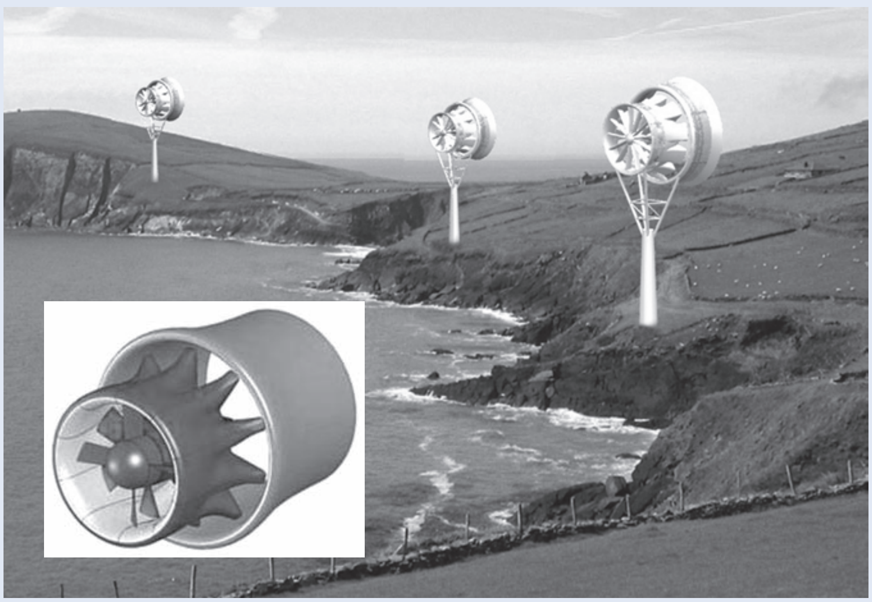
8.13 Summary and Useful Equations

Case Study in Energy and the Environment

Wind Power: *The FloDesign Wind Turbine*

We are all now familiar with the ubiquitous three-bladed wind turbines that are being used to generate increasing amounts of power. The technology is already quite mature, so new developments will be incremental: improved blade designs, better controls, and composite materials to allow larger turbines. The largest in the world, being built by a Norwegian team, will be 533 ft tall with a rotor diameter of 475 ft, and it will generate about 10 MW, sufficient for more than 2000 homes. Bearing in mind that the Empire State Building is 1250 ft tall, this wind turbine will be huge—so big it must be installed offshore.

Engineers are still investigating alternatives to these designs. *FloDesign Wind Turbine*, a spin-off from the aerospace company *FloDesign* based in Wilbraham, Massachusetts, is developing a prototype that, according to CEO Stanley Kowalski III, will be up to three times more efficient than conventional wind turbines. From the front, the wind turbine looks something like the air intake of a jet engine (not surprisingly, considering *FloDesign*'s history). The shaped cowlings shown in the figure guide the air into spinning vortices as it exits the device, accelerating the flow and causing a significant pressure drop. The incoming wind first meets a set of fixed stator blades, which direct it onto the rotor blades to extract power from the flow. The exiting air hence has lower energy and velocity than the air flowing around the turbine, but the device's shroud is so shaped that the relatively fast-moving outside air is blended with the exiting air in the area just behind the rotors, creating a low-pressure region behind the turbine blades. This is where the device has an advantage over conventional turbines; the induced low-pressure region actually draws air into the device at an increased rate, generating more power. This idea is not new, but past attempts to build similar turbines were limited by the fact that such a turbine had to be very precisely aligned with the wind's direction (within



Two views of the FloDesign Wind Turbine (Pictures courtesy of FloDesign Wind Turbine)

about 4°); this device will work at angles of up to 20° off the wind.

Theoretically (as we'll learn in Chapter 10), conventional wind turbines capture a maximum of 59.3 percent of the wind energy. The new design generates as much power as a conventional wind turbine with blades twice as big. The smaller blade size of the new design means the *FloDesign Wind Turbine* could be packed closer together than conventional turbines, increasing the amount of power that can be generated per acre of land. Because its blades are lighter and smaller, the design starts spinning and generating power at lower wind speeds, and it is more tolerant of unstable wind patterns, making it excellent for windy regions where large turbines cannot be used, such as in cities. Smaller blades can also be allowed to spin faster, reducing the need for expensive gearboxes that conventional wind turbines must use to connect slow-moving rotors to a high-speed generator. With fewer gears and other moving parts, the company claims it can reduce the number of components by up to 75 percent, reducing costs and making maintenance easier.

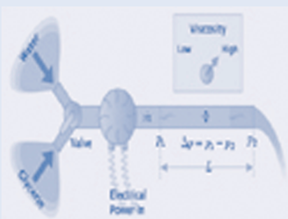
FloDesign has already built a small prototype for wind-tunnel tests. Their next step is to build a 12-ft-diameter, 10-kW system for field tests. The prototype will be finished in 2010, with commercial wind turbines to follow.

VIDEO

The Reynolds Transition Experiment.

VIDEO

Variable Viscosity Experiment (Animation).



Flows completely bounded by solid surfaces are called internal flows. Thus internal flows include many important and practical flows such as those through pipes, ducts, nozzles, diffusers, sudden contractions and expansions, valves, and fittings.

Internal flows may be laminar or turbulent. Some laminar flow cases may be solved analytically. In the case of turbulent flow, analytical solutions are not possible, and we must rely heavily on semi-empirical theories and on experimental data. The nature of laminar and turbulent flows was discussed in Section 2.6. For internal flows, the flow regime (laminar or turbulent) is primarily a function of the Reynolds number.

In this chapter we will only consider incompressible flows; hence we will study the flow of liquids as well as gases that have negligible heat transfer and for which the Mach number $M < 0.3$; a value of $M = 0.3$ in air corresponds to a speed of approximately 100 m/s. Following a brief introduction, this chapter is divided into the following parts:

Part A: Part A discusses fully developed laminar flow of a Newtonian fluid between parallel plates and in a pipe. These two cases can be studied analytically.

Part B: Part B is about laminar and turbulent flows in pipes and ducts. The laminar flow analysis follows from Part A; the turbulent flow (which is the most common) is too complex to be analyzed, so experimental data will be used to develop solution techniques.

Part C: Part C is a discussion of methods of flow measurement.

8.1 Introduction

Laminar versus Turbulent Flow

VIDEO

Variable Viscosity Experiment: Pressure Drop.

As discussed previously in Section 2.6, the pipe flow regime (laminar or turbulent) is determined by the Reynolds number, $Re = \rho \overline{V} D/\mu$. One can demonstrate, by the classic Reynolds experiment, the qualitative difference between laminar and turbulent flows. In this experiment water flows from a large reservoir through a clear tube. A thin filament of dye injected at the entrance to the tube allows visual observation of the flow. At low flow rates (low Reynolds numbers) the dye injected into the flow remains in a single filament along the tube; there is little dispersion of dye because the flow is laminar. A laminar flow is one in which the fluid flows in laminae, or layers; there is no macroscopic mixing of adjacent fluid layers.

As the flow rate through the tube is increased, the dye filament eventually becomes unstable and breaks up into a random motion throughout the tube; the line of dye is stretched and twisted into myriad entangled threads, and it quickly disperses throughout the entire flow field. This behavior of turbulent flow is caused by small, high-frequency velocity fluctuations superimposed on the mean motion of a turbulent flow, as illustrated earlier in Fig. 2.15; the mixing of fluid particles from adjacent layers of fluid results in rapid dispersion of the dye. We mentioned in Chapter 2 an everyday example of the difference between laminar and turbulent flow—when you gently turn on the kitchen faucet (not aerated). For very low flow rates, the water exits smoothly (indicating laminar flow in the pipe); for higher flow rates, the flow is churned up (turbulent flow).

Under normal conditions, transition to turbulence occurs at $Re \approx 2300$ for flow in pipes: For water flow in a 1-in. diameter pipe, this corresponds to an average speed of 0.3 ft/s. With great care to maintain the flow free from disturbances, and with smooth surfaces, experiments have been able to maintain laminar flow in a pipe to a Reynolds number of about 100,000! However, most engineering flow situations are not so carefully controlled, so we will take $Re \approx 2300$ as our benchmark for transition to turbulence. Transition Reynolds numbers for some other flow situations are given in the Examples. Turbulence occurs when the viscous forces in the fluid are unable to damp out random fluctuations in the fluid motion (generated, for example, by roughness of a pipe wall), and the flow becomes chaotic. For example, a high-viscosity fluid such as motor oil is able to damp out fluctuations more effectively than a low viscosity fluid such as water and therefore remains laminar even at relatively high flow rates. On the other hand, a high-density fluid will generate significant inertia forces due to the random fluctuations in the motion, and this fluid will transition to turbulence at a relatively low flow rate.

The Entrance Region

Figure 8.1 illustrates laminar flow in the entrance region of a circular pipe. The flow has uniform velocity U_0 at the pipe entrance. Because of the no-slip condition at the wall, we know that the velocity at the wall must be zero along the entire length of the pipe. A boundary layer (Section 2.6) develops along the walls of the channel. The solid surface exerts a retarding shear force on the flow; thus the speed of the fluid in the neighborhood of the surface is reduced. At successive sections along the pipe in this entry region, the effect of the solid surface is felt farther out into the flow.

For incompressible flow, mass conservation requires that, as the speed close to the wall is reduced, the speed in the central frictionless region of the pipe must increase slightly to compensate; for this inviscid central region, then, the pressure (as indicated by the Bernoulli equation) must also drop somewhat.

Sufficiently far from the pipe entrance, the boundary layer developing on the pipe wall reaches the pipe centerline and the flow becomes entirely viscous. The velocity profile shape then changes slightly after the inviscid core disappears. When the profile shape no longer changes with increasing distance x, the flow is called *fully developed*. The distance

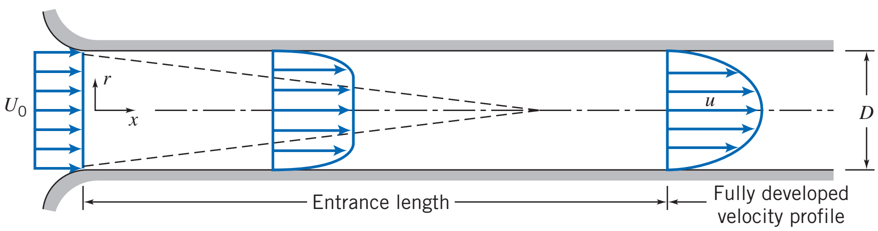


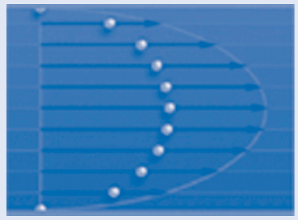
Fig. 8.1 Flow in the entrance region of a pipe.

CLASSIC VIDEO

Turbulence.

VIDEO

Laminar Pipe Flow: Velocity Profile.

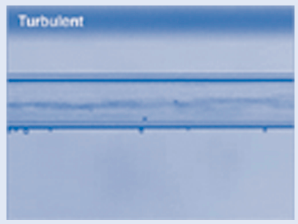


VIDEO

Pipe Flow: Laminar.

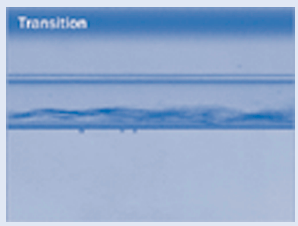
VIDEO

Pipe Flow: Transitional.



VIDEO

Pipe Flow: Turbulent.



downstream from the entrance to the location at which fully developed flow begins is called the *entrance length*. The actual shape of the fully developed velocity profile depends on whether the flow is laminar or turbulent. In Fig. 8.1 the profile is shown qualitatively for a laminar flow. Although the velocity profiles for some fully developed laminar flows can be obtained by simplifying the complete equations of motion from Chapter 5, turbulent flows cannot be so treated.

CLASSIC VIDEO

Turbulence.

For laminar flow, it turns out that entrance length, L, is a function of Reynolds number,

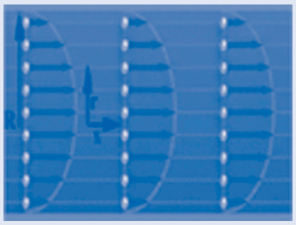
$$\frac{L}{D} \simeq 0.06 \frac{\rho \overline{V} D}{\mu} \tag{8.1}$$

where $\overline{V} \equiv Q/A$ is the average velocity (because flow rate $Q = A\overline{V} = AU_0$, we have $\overline{V} = U_0$). Laminar flow in a pipe may be expected only for Reynolds numbers less than 2300. Thus the entrance length for laminar pipe flow may be as long as

$$L \simeq 0.06\, ReD \leq (0.06)(2300)\, D = 138D$$

VIDEO

Fully-Developed Pipe Flow.



or nearly 140 pipe diameters. If the flow is turbulent, enhanced mixing among fluid layers causes more rapid growth of the boundary layer. Experiments show that the mean velocity profile becomes fully developed within 25 to 40 pipe diameters from the entrance. However, the details of the turbulent motion may not be fully developed for 80 or more pipe diameters. We are now ready to study laminar internal flows (Part A), as well as laminar and turbulent flows in pipes and ducts (Part B). For these we will be focusing on what happens after the entrance region, i.e., fully developed flows.

Part A Fully Developed Laminar Flow

In this section we consider a few classic examples of fully developed laminar flows. Our intent is to obtain detailed information about the velocity field because knowledge of the velocity field permits calculation of shear stress, pressure drop, and flow rate.

8.2 Fully Developed Laminar Flow Between Infinite Parallel Plates

The flow between parallel plates is appealing because the geometry is the simplest possible, but why *would* there be a flow at all? The answer is that flow could be generated by applying a pressure gradient parallel to the plates, or by moving one plate parallel with respect to the other, or by having a body force (e.g., gravity) parallel to the plates, or by a combination of these driving mechanisms. We will consider all of these possibilities.

Both Plates Stationary

Fluid in high-pressure hydraulic systems (such as the brake system of an automobile) often leaks through the annular gap between a piston and cylinder. For very small gaps (typically 0.005 mm or less), this flow field may be modeled as flow between infinite parallel plates, as indicated in the sketch of Fig. 8.2. To calculate the leakage flow rate, we must first determine the velocity field.

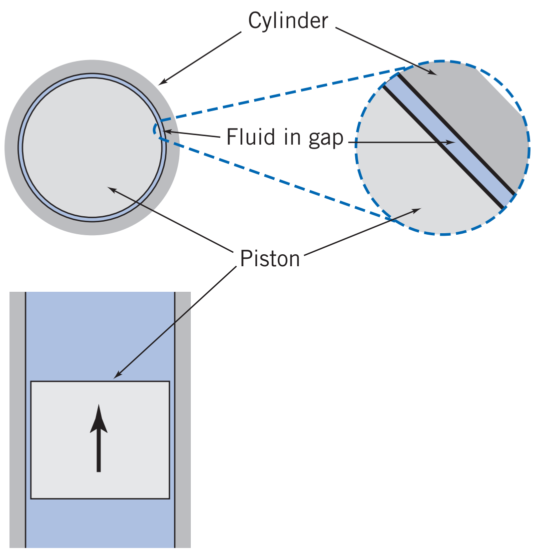


Fig. 8.2 Piston-cylinder approximated as parallel plates.

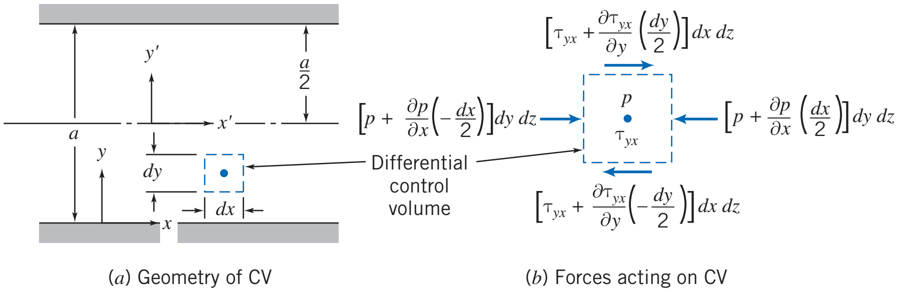


Fig. 8.3 Control volume for analysis of laminar flow between stationary infinite parallel plates.

Let us consider the fully developed laminar flow between horizontal infinite parallel plates. The plates are separated by distance a, as shown in Fig. 8.3. The plates are considered infinite in the z direction, with no variation of any fluid property in this direction. The flow is also assumed to be steady and incompressible. Before starting our analysis, what do we know about the flow field? For one thing we know that the x component of velocity must be zero at both the upper and lower plates as a result of the no-slip condition at the wall. The boundary conditions are then

$$\begin{aligned} &\text{at} \quad y = 0 \qquad u = 0 \\ &\text{at} \quad y = a \qquad u = 0 \end{aligned}$$

Since the flow is fully developed, the velocity cannot vary with x and, hence, depends on y only, so that $u = u(y)$. Furthermore, there is no component of velocity in either the y or z direction ($v = w = 0$). In fact, for fully developed flow only the pressure can and will change (in a manner to be determined from the analysis) in the x direction.

This is an obvious case for using the Navier–Stokes equations in rectangular coordinates (Eqs. 5.27). Using the above assumptions, these equations can be greatly simplified and then solved using the boundary conditions (see Problem 8.17). In this section we will instead take a longer route—using a differential control volume—to bring out some important features of the fluid mechanics.

For our analysis we select a differential control volume of size $d\Psi = dx\,dy\,dz$, and apply the x component of the momentum equation.

Basic equation:

$$F_{S_x} + \underbrace{\cancel{F_{B_x}}}_{=0(3)} = \underbrace{\cancel{\frac{\partial}{\partial t}}}_{=0(1)} \int_{\mathrm{CV}} u\,\rho\,d\forall + \int_{\mathrm{CS}} u\,\rho \vec{V}\cdot d\vec{A} \tag{4.18a}$$

Assumptions: (1) Steady flow (given)
(2) Fully developed flow (given)
(3) $F_{B_x} = 0$ (given)

The very nature of fully developed flow is that the velocity profile is the same at all locations along the flow; hence there is no change in momentum. Equation 4.18a then reduces to the simple result that the sum of the surface forces on the control volume is zero,

$$F_{S_x} = 0 \tag{8.2}$$

The next step is to sum the forces acting on the control volume in the x direction. We recognize that normal forces (pressure forces) act on the left and right faces and tangential forces (shear forces) act on the top and bottom faces.

If the pressure at the center of the element is p, then the pressure force on the left face is

$$dF_L = \left(p - \frac{\partial p}{\partial x}\frac{dx}{2}\right) dy\,dz$$

and the pressure force on the right face is

$$dF_R = -\left(p + \frac{\partial p}{\partial x}\frac{dx}{2}\right) dy\,dz$$

If the shear stress at the center of the element is τ_{yx}, then the shear force on the bottom face is

$$dF_B = -\left(\tau_{yx} - \frac{d\tau_{yx}}{dy}\frac{dy}{2}\right) dx\,dz$$

and the shear force on the top face is

$$dF_T = \left(\tau_{yx} + \frac{d\tau_{yx}}{dy}\frac{dy}{2}\right) dx\,dz$$

Note that in expanding the shear stress, τ_{yx}, in a Taylor series about the center of the element, we have used the total derivative rather than a partial derivative. We did this because we recognized that τ_{yx} is only a function of y, since $u = u(y)$.

Using the four surface forces dF_L, dF_R, dF_B, and dF_T in Eq. 8.2, this equation simplifies to

$$\frac{\partial p}{\partial x} = \frac{d\tau_{yx}}{dy} \tag{8.3}$$

This equation states that because there is no change in momentum, the net pressure force (which is actually $-\partial p/\partial x$) balances the net friction force (which is actually $-d\tau_{yx}/dy$). Equation 8.3 has an interesting feature: The left side is at most a function of x only (this follows immediately from writing the y component of the momentum equation); the right side is at most a function of y only (the flow is fully developed, so

it does not change with x). Hence, the only way the equation can be valid for all x and y is for each side to in fact be constant:

$$\frac{d\tau_{yx}}{dy} = \frac{\partial p}{\partial x} = \text{constant}$$

Integrating this equation, we obtain

$$\tau_{yx} = \left(\frac{\partial p}{\partial x}\right) y + c_1$$

which indicates that the shear stress varies linearly with y. We wish to find the velocity distribution. To do so, we need to relate the shear stress to the velocity field. For a Newtonian fluid we can use Eq. 2.15 because we have a one-dimensional flow [or we could have started with the full stress equation (Eq. 5.25a) and simplified],

$$\tau_{yx} = \mu \frac{du}{dy} \tag{2.15}$$

so we get

$$\mu \frac{du}{dy} = \left(\frac{\partial p}{\partial x}\right) y + c_1$$

Integrating again

$$u = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right) y^2 + \frac{c_1}{\mu} y + c_2 \tag{8.4}$$

It is interesting to note that if we had started with the Navier–Stokes equations (Eqs. 5.27) instead of using a differential control volume, after only a few steps (i.e., simplifying and integrating twice) we would have obtained Eq. 8.4 (see Problem 8.17). To evaluate the constants, c_1 and c_2, we must apply the boundary conditions. At $y = 0$, $u = 0$. Consequently, $c_2 = 0$. At $y = a$, $u = 0$. Hence

$$0 = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right) a^2 + \frac{c_1}{\mu} a$$

This gives

$$c_1 = -\frac{1}{2}\left(\frac{\partial p}{\partial x}\right) a$$

and hence,

$$u = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right) y^2 - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right) ay = \frac{a^2}{2\mu}\left(\frac{\partial p}{\partial x}\right)\left[\left(\frac{y}{a}\right)^2 - \left(\frac{y}{a}\right)\right] \tag{8.5}$$

At this point we have the velocity profile. This is key to finding other flow properties, as we next discuss.

Shear Stress Distribution

The shear stress distribution is given by

$$\tau_{yx} = \left(\frac{\partial p}{\partial x}\right) y + c_1 = \left(\frac{\partial p}{\partial x}\right) y - \frac{1}{2}\left(\frac{\partial p}{\partial x}\right) a = a\left(\frac{\partial p}{\partial x}\right)\left[\frac{y}{a} - \frac{1}{2}\right] \tag{8.6a}$$

Volume Flow Rate

The volume flow rate is given by

$$Q = \int_A \vec{V} \cdot d\vec{A}$$

For a depth l in the z direction,

$$Q = \int_0^a ul\, dy \qquad \text{or} \qquad \frac{Q}{l} = \int_0^a \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - ay)\, dy$$

Thus the volume flow rate per unit depth is given by

$$\frac{Q}{l} = -\frac{1}{12\mu}\left(\frac{\partial p}{\partial x}\right)a^3 \tag{8.6b}$$

Flow Rate as a Function of Pressure Drop

Since $\partial p/\partial x$ is constant, the pressure varies linearly with x and

$$\frac{\partial p}{\partial x} = \frac{p_2 - p_1}{L} = \frac{-\Delta p}{L}$$

Substituting into the expression for volume flow rate gives

$$\frac{Q}{l} = -\frac{1}{12\mu}\left[\frac{-\Delta p}{L}\right]a^3 = \frac{a^3 \Delta p}{12\mu L} \tag{8.6c}$$

Average Velocity

The average velocity magnitude, $\overline{V}$, is given by

$$\overline{V} = \frac{Q}{A} = -\frac{1}{12\mu}\left(\frac{\partial p}{\partial x}\right)\frac{a^3 l}{la} = -\frac{1}{12\mu}\left(\frac{\partial p}{\partial x}\right)a^2 \tag{8.6d}$$

Point of Maximum Velocity

To find the point of maximum velocity, we set du/dy equal to zero and solve for the corresponding y. From Eq. 8.5

$$\frac{du}{dy} = \frac{a^2}{2\mu}\left(\frac{\partial p}{\partial x}\right)\left[\frac{2y}{a^2} - \frac{1}{a}\right]$$

Thus,

$$\frac{du}{dy} = 0 \qquad \text{at} \qquad y = \frac{a}{2}$$

At

$$y = \frac{a}{2}, \qquad u = u_{\max} = -\frac{1}{8\mu}\left(\frac{\partial p}{\partial x}\right)a^2 = \frac{3}{2}\overline{V} \tag{8.6c}$$

Transformation of Coordinates

In deriving the above relations, the origin of coordinates, $y = 0$, was taken at the bottom plate. We could just as easily have taken the origin at the centerline of the channel. If we

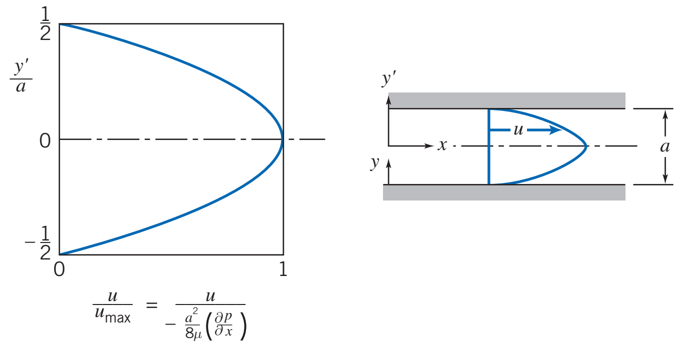


Fig. 8.4 Dimensionless velocity profile for fully developed laminar flow between infinite parallel plates.

denote the coordinates with origin at the channel centerline as x, y', the boundary conditions are $u = 0$ at $y' = \pm a/2$.

To obtain the velocity profile in terms of x, y', we substitute $y = y' + a/2$ into Eq. 8.5. The result is

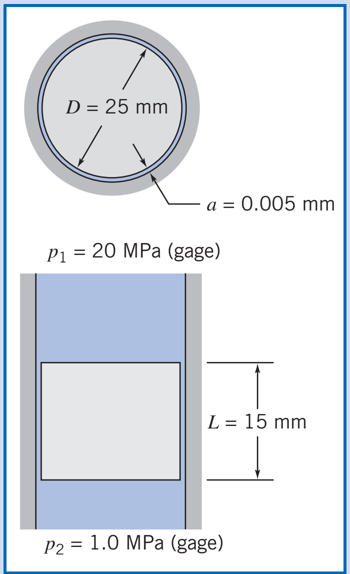
$$u = \frac{a^2}{2\mu}\left(\frac{\partial p}{\partial x}\right)\left[\left(\frac{y'}{a}\right)^2 - \frac{1}{4}\right] \tag{8.7}$$

Equation 8.7 shows that the velocity profile for laminar flow between stationary parallel plates is parabolic, as shown in Fig. 8.4.

Since all stresses were related to velocity gradients through Newton's law of viscosity, and the additional stresses that arise as a result of turbulent fluctuations have not been accounted for, *all of the results in this section are valid for laminar flow only*. Experiments show that laminar flow between stationary parallel plates becomes turbulent for Reynolds numbers (defined as $Re = \rho \vec{V} a/\mu$) greater than approximately 1400. Consequently, the Reynolds number should be checked after using Eqs. 8.6 to ensure a valid solution.

Example 8.1 LEAKAGE FLOW PAST A PISTON

A hydraulic system operates at a gage pressure of 20 MPa and 55°C. The hydraulic fluid is SAE 10W oil. A control valve consists of a piston 25 mm in diameter, fitted to a cylinder with a mean radial clearance of 0.005 mm. Determine the leakage flow rate if the gage pressure on the low-pressure side of the piston is 1.0 MPa. (The piston is 15 mm long.)



Given: Flow of hydraulic oil between piston and cylinder, as shown. Fluid is SAE 10W oil at 55°C.

Find: Leakage flow rate, Q.

Solution:
The gap width is very small, so the flow may be modeled as flow between parallel plates. Equation 8.6c may be applied.

Governing equation:

$$\frac{Q}{l} = \frac{a^3 \Delta p}{12\mu L} \tag{8.6c}$$

Assumptions: (1) Laminar flow.
(2) Steady flow.
(3) Incompressible flow.
(4) Fully developed flow.
(Note $L/a = 15/0.005 = 3000!$)

The plate width, l, is approximated as $l = \pi D$. Thus

$$Q = \frac{\pi D a^3 \Delta p}{12\mu L}$$

For SAE 10W oil at 55°C, $\mu = 0.018\ \mathrm{kg/(m \cdot s)}$, from Fig. A.2, Appendix A. Thus

$$Q = \frac{\pi}{12} \times 25\ \mathrm{mm} \times (0.005)^3\ \mathrm{mm}^3 \times (20-1)10^6\ \frac{\mathrm{N}}{\mathrm{m}^2} \times \frac{\mathrm{m \cdot s}}{0.018\ \mathrm{kg}} \times \frac{1}{15\ \mathrm{mm}} \times \frac{\mathrm{kg \cdot m}}{\mathrm{N \cdot s^2}}$$

$$Q = 57.6\ \mathrm{mm}^3/\mathrm{s} \qquad \longleftarrow \qquad Q$$

To ensure that flow is laminar, we also should check the Reynolds number.

$$\overline{V} = \frac{Q}{A} = \frac{Q}{\pi D a} = 57.6\ \frac{\mathrm{mm}^3}{\mathrm{s}} \times \frac{1}{\pi} \times \frac{1}{25\ \mathrm{mm}} \times \frac{1}{0.005\ \mathrm{mm}} \times \frac{\mathrm{m}}{10^3\ \mathrm{mm}} = 0.147\ \mathrm{m/s}$$

and

$$Re = \frac{\rho \overline{V} a}{\mu} = \frac{SG \rho_{H_2O} \overline{V} a}{\mu}$$

For SAE 10W oil, SG = 0.92, from Table A.2, Appendix A. Thus

$$Re = 0.92 \times 1000\frac{\mathrm{kg}}{\mathrm{m}^3} \times 0.147\frac{\mathrm{m}}{\mathrm{s}} \times 0.005\ \mathrm{mm} \times \frac{\mathrm{m \cdot s}}{0.018\ \mathrm{kg}} \times \frac{\mathrm{m}}{10^3\ \mathrm{mm}} = 0.0375$$

Thus flow is surely laminar, since $Re \ll 1400$.

Upper Plate Moving with Constant Speed, *U*

The second basic way to generate flow between infinite parallel plates is to have one plate move parallel to the other, either with or without an applied pressure gradient. We will next analyze this problem for the case of laminar flow.

Such a flow commonly occurs, for example, in a journal bearing (a commonly used type of bearing, e.g., the main crankshaft bearings in the engine of an automobile). In such a bearing, an inner cylinder, the journal, rotates inside a stationary member. At light loads, the centers of the two members essentially coincide, and the small clearance gap is symmetric. Since the gap is small, it is reasonable to "unfold" the bearing and to model the flow field as flow between infinite parallel plates, as indicated in the sketch of Fig. 8.5.

Let us now consider a case where the upper plate is moving to the right with constant speed, U. All we have done in going from a stationary upper plate to a moving upper plate is to change one of the boundary conditions. The boundary conditions for the moving plate case are

$$u = 0 \quad \text{at} \quad y = 0$$
$$u = U \quad \text{at} \quad y = a$$

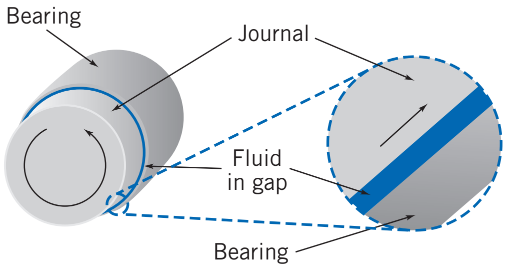


Fig. 8.5 Journal bearing approximated as parallel plates.

Since only the boundary conditions have changed, there is no need to repeat the entire analysis of the previous section. The analysis leading to Eq. 8.4 is equally valid for the moving plate case. Thus the velocity distribution is given by

$$u = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)y^2 + \frac{c_1}{\mu}\,y + c_2 \tag{8.4}$$

and our only task is to evaluate constants c_1 and c_2 by using the appropriate boundary conditions. [Note once again that using the full Navier–Stokes equations (Eqs. 5.27) would have led very quickly to Eq. 8.4.]

At $y = 0$, $u = 0$. Consequently, $c_2 = 0$.
At $y = a$, $u = U$. Consequently,

$$U = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)a^2 + \frac{c_1}{\mu}\,a \qquad \text{and thus} \qquad c_1 = \frac{U\mu}{a} - \frac{1}{2}\left(\frac{\partial p}{\partial x}\right)a$$

Hence,

$$u = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)y^2 + \frac{Uy}{a} - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)ay = \frac{Uy}{a} + \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - ay)$$

$$u = \frac{Uy}{a} + \frac{a^2}{2\mu}\left(\frac{\partial p}{\partial x}\right)\left[\left(\frac{y}{a}\right)^2 - \left(\frac{y}{a}\right)\right] \tag{8.8}$$

It is reassuring to note that Eq. 8.8 reduces to Eq. 8.5 for a stationary upper plate (set $U = 0$). From Eq. 8.8, for zero pressure gradient (for $\partial p/\partial x = 0$) the velocity varies linearly with y. This was the case treated earlier in Chapter 2; this linear profile is called a *Couette* flow, after a 19th-century physicist.

We can obtain additional information about the flow from the velocity distribution of Eq. 8.8.

Shear Stress Distribution

The shear stress distribution is given by $\tau_{yx} = \mu(du/dy)$,

$$\tau_{yx} = \mu\,\frac{U}{a} + \frac{a^2}{2}\left(\frac{\partial p}{\partial x}\right)\left[\frac{2y}{a^2} - \frac{1}{a}\right] = \mu\,\frac{U}{a} + a\left(\frac{\partial p}{\partial x}\right)\left[\frac{y}{a} - \frac{1}{2}\right] \tag{8.9a}$$

Volume Flow Rate

The volume flow rate is given by $Q = \int_A \vec{V}\cdot d\vec{A}$. For depth l in the z direction

$$Q = \int_0^a ul\,dy \qquad \text{or} \qquad \frac{Q}{l} = \int_0^a \left[\frac{Uy}{a} + \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - ay)\right]dy$$

Thus the volume flow rate per unit depth is given by

$$\frac{Q}{l} = \frac{Ua}{2} - \frac{1}{12\mu}\left(\frac{\partial p}{\partial x}\right)a^3 \tag{8.9b}$$

Average Velocity

The average velocity magnitude, $\overline{V}$, is given by

$$\overline{V} = \frac{Q}{A} = l\left[\frac{Ua}{2} - \frac{1}{12\mu}\left(\frac{\partial p}{\partial x}\right)a^3\right]\Big/ la = \frac{U}{2} - \frac{1}{12\mu}\left(\frac{\partial p}{\partial x}\right)a^2 \tag{8.9c}$$

Point of Maximum Velocity

To find the point of maximum velocity, we set du/dy equal to zero and solve for the corresponding y. From Eq. 8.8

$$\frac{du}{dy} = \frac{U}{a} + \frac{a^2}{2\mu}\left(\frac{\partial p}{\partial x}\right)\left[\frac{2y}{a^2} - \frac{1}{a}\right] = \frac{U}{a} + \frac{a}{2\mu}\left(\frac{\partial p}{\partial x}\right)\left[2\left(\frac{y}{a}\right) - 1\right]$$

Thus,

$$\frac{du}{dy} = 0 \qquad \text{at} \qquad y = \frac{a}{2} - \frac{U/a}{(1/\mu)(\partial p/\partial x)}$$

There is no simple relation between the maximum velocity, u_{max}, and the mean velocity, $\overline{V}$, for this flow case.

Equation 8.8 suggests that the velocity profile may be treated as a combination of a linear and a parabolic velocity profile; the last term in Eq. 8.8 is identical to that in Eq. 8.5. The result is a family of velocity profiles, depending on U and $(1/\mu)(\partial p/\partial x)$; three profiles are sketched in Fig. 8.6. (As shown in Fig. 8.6, some reverse flow—flow in the negative x direction—can occur when $\partial p/\partial x > 0$.)

Again, all of the results developed in this section are valid for laminar flow only. Experiments show that this flow becomes turbulent (for $\partial p/\partial x = 0$) at a Reynolds number of approximately 1500, where $Re = \rho Ua/\mu$ for this flow case. Not much information is available for the case where the pressure gradient is not zero.

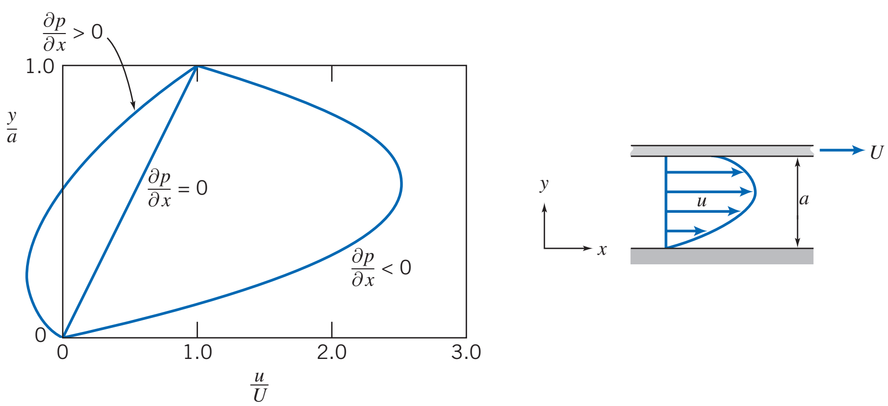
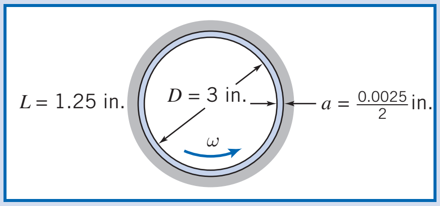


Fig. 8.6 Dimensionless velocity profile for fully developed laminar flow between infinite parallel plates: upper plate moving with constant speed, U.

Example 8.2 TORQUE AND POWER IN A JOURNAL BEARING

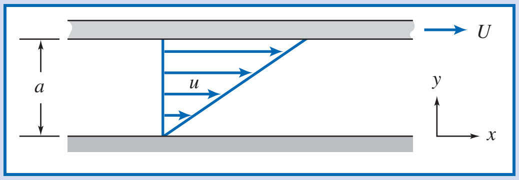
A crankshaft journal bearing in an automobile engine is lubricated by SAE 30 oil at 210°F. The bearing diameter is 3 in., the diametral clearance is 0.0025 in., and the shaft rotates at 3600 rpm; it is 1.25 in. long. The bearing is under no load, so the clearance is symmetric. Determine the torque required to turn the journal and the power dissipated.

Given: Journal bearing, as shown. Note that the gap width, a, is *half* the diametral clearance. Lubricant is SAE 30 oil at 210°F. Speed is 3600 rpm.



Find: (a) Torque, T.
(b) Power dissipated.

Solution:
Torque on the journal is caused by viscous shear in the oil film. The gap width is small, so the flow may be modeled as flow between infinite parallel plates:



Governing equation:

$$\tau_{yx} = \mu \frac{U}{a} + a \underbrace{\left(\frac{\partial p}{\partial x}\right)}_{=0(6)} \left[\frac{y}{a} - \frac{1}{2}\right] \tag{8.9a}$$

Assumptions:
(1) Laminar flow.
(2) Steady flow.
(3) Incompressible flow.
(4) Fully developed flow.
(5) Infinite width ($L/a = 1.25/0.00125 = 1000$, so this is a reasonable assumption).
(6) $\partial p/\partial x = 0$ (flow is symmetric in the actual bearing at no load).

Then

$$\tau_{yx} = \mu \frac{U}{a} = \mu \frac{\omega R}{a} = \mu \frac{\omega D}{2a}$$

For SAE 30 oil at 210°F (99°C), $\mu = 9.6 \times 10^{-3}\,\text{N} \cdot \text{s/m}^2 (2.01 \times 10^{-4}\ \text{lbf} \cdot \text{s/ft}^2)$, from Fig. A.2, Appendix A. Thus,

$$\tau_{yx} = 2.01 \times 10^{-4} \frac{\text{lbf} \cdot \text{s}}{\text{ft}^2} \times 3600 \frac{\text{rev}}{\text{min}} \times 2\pi \frac{\text{rad}}{\text{rev}} \times \frac{\text{min}}{60\ \text{s}} \times 3\ \text{in.} \times \frac{1}{2} \times \frac{1}{0.00125\ \text{in.}}$$

$$\tau_{yx} = 90.9\ \text{lbf/ft}^2$$

The total shear force is given by the shear stress times the area. It is applied to the journal surface. Therefore, for the torque

$$T = FR = \tau_{yx} \pi D L R = \frac{\pi}{2} \tau_{yx} D^2 L$$

$$= \frac{\pi}{2} \times 90.9 \frac{\text{lbf}}{\text{ft}^2} \times (3)^2\ \text{in.}^2 \times \frac{\text{ft}^2}{144\ \text{in.}^2} \times 1.25\ \text{in.}$$

$$T = 11.2\ \text{in.} \cdot \text{lbf} \longleftarrow T$$

The power dissipated in the bearing is

$$\dot{W} = FU = FR\omega = T\omega$$
$$= 11.2\,\text{in.}\cdot\text{lbf}\times 3600\,\frac{\text{rev}}{\text{min}}\times\frac{\text{min}}{60\,\text{s}}\times 2\pi\,\frac{\text{rad}}{\text{rev}}\times\frac{\text{ft}}{12\,\text{in.}}\times\frac{\text{hp}\cdot\text{s}}{550\,\text{ft}\cdot\text{lbf}}$$

$$\dot{W} = 0.640\,\text{hp} \longleftarrow \dot{W}$$

To ensure laminar flow, check the Reynolds number.

$$Re = \frac{\rho U a}{\mu} = \frac{SG\rho_{H_2O}Ua}{\mu} = \frac{SG\rho_{H_2O}\omega Ra}{\mu}$$

Assume, as an approximation, the specific gravity of SAE 30 oil is the same as that of SAE 10W oil. From Table A.2, Appendix A, SG = 0.92. Thus

$$Re = 0.92\times 1.94\,\frac{\text{slug}}{\text{ft}^3}\times\frac{(3600)2\pi}{60}\,\frac{\text{rad}}{\text{s}}\times 1.5\,\text{in.}\times 0.00125\,\text{in.}$$
$$\times\frac{\text{ft}^2}{2.01\times10^{-4}\,\text{lbf}\cdot\text{s}}\times\frac{\text{ft}^2}{144\,\text{in.}^2}\times\frac{\text{lbf}\cdot\text{s}^2}{\text{slug}\cdot\text{ft}}$$

$$Re = 43.6$$

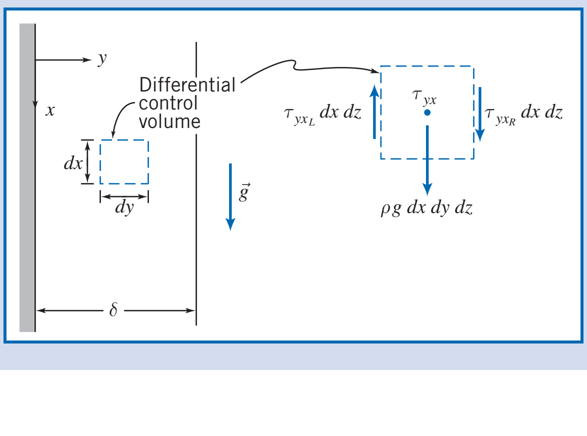
Therefore, the flow is laminar, since $Re \ll 1500$.

In this problem we approximated the circular-streamline flow in a small annular gap as a linear flow between infinite parallel plates. As we saw in Example 5.10, for the small value of the gap width a to radius R ratio a/R (in this problem <1%), the error in shear stress is about $\frac{1}{2}$ of this ratio. Hence, the error introduced is insignificant—much less than the uncertainty associated with obtaining a viscosity for the oil.

We have seen how steady, one-dimensional laminar flows between two plates can be generated by applying a pressure gradient, by moving one plate with respect to the other, or by having both driving mechanisms present. To finish our discussion of this type of flow, Example 8.3 examines a *gravity-driven* steady, one-dimensional laminar flow down a vertical wall. Once again, the direct approach would be to start with the two-dimensional rectangular coordinate form of the Navier–Stokes equations (Eqs. 5.27; see Problem 8.44); instead we will use a differential control volume.

Example 8.3 LAMINAR FILM ON A VERTICAL WALL

A viscous, incompressible, Newtonian liquid flows in steady, laminar flow down a vertical wall. The thickness, δ, of the liquid film is constant. Since the liquid free surface is exposed to atmospheric pressure, there is no pressure gradient. For this gravity-driven flow, apply the momentum equation to differential control volume $dx\,dy\,dz$ to derive the velocity distribution in the liquid film.



Given: Fully developed laminar flow of incompressible, Newtonian liquid down a vertical wall; thickness, δ, of the liquid film is constant and $\partial p/\partial x = 0$.

Find: Expression for the velocity distribution in the film.

Solution:

The x component of the momentum equation for a control volume is

$$F_{S_x} + F_{B_x} = \frac{\partial}{\partial t}\int_{CV} u\,\rho\,d\forall + \int_{CS} u\,\rho\vec{V}\cdot d\vec{A} \qquad (4.18a)$$

Under the conditions given we are dealing with a steady, incompressible, fully developed laminar flow.

For steady flow, $\dfrac{\partial}{\partial t}\displaystyle\int_{CV} u\,\rho\,d\forall = 0$

For fully developed flow, $\displaystyle\int_{CS} u\,\rho\vec{V}\cdot d\vec{A} = 0$

Thus the momentum equation for the present case reduces to

$$F_{S_x} + F_{B_x} = 0$$

The body force, F_{B_x}, is given by $F_{B_x} = \rho g\, d\forall = \rho g\, dx\, dy\, dz$. The only surface forces acting on the differential control volume are shear forces on the vertical surfaces. (Since we have a free-surface flow, with straight streamlines, the pressure is atmospheric throughout; no net pressure forces act on the control volume.)

If the shear stress at the center of the differential control volume is τ_{yx}, then,

$$\text{shear stress on left face is } \tau_{yx_L} = \left(\tau_{yx} - \frac{d\tau_{yx}}{dy}\frac{dy}{2}\right)$$

and

$$\text{shear stress on right face is } \tau_{yx_R} = \left(\tau_{yx} + \frac{d\tau_{yx}}{dy}\frac{dy}{2}\right)$$

The direction of the shear stress vectors is taken consistent with the sign convention of Section 2.3. Thus on the left face, a minus y surface, τ_{yx_L} acts upward, and on the right face, a plus y surface, τ_{yx_R} acts downward.

The surface forces are obtained by multiplying each shear stress by the area over which it acts. Substituting into $F_{S_x} + F_{B_x} = 0$, we obtain

$$-\tau_{yx_L}\, dx\, dz + \tau_{yx_R}\, dx\, dz + \rho g\, dx\, dy\, dz = 0$$

or

$$-\left(\tau_{yx} - \frac{d\tau_{yx}}{dy}\frac{dy}{2}\right) dx\, dz + \left(\tau_{yx} + \frac{d\tau_{yx}}{dy}\frac{dy}{2}\right) dx\, dz + \rho g\, dx\, dy\, dz = 0$$

Simplifying gives

$$\frac{d\tau_{yx}}{dy} + \rho g = 0 \quad \text{or} \quad \frac{d\tau_{yx}}{dy} = -\rho g$$

Since

$$\tau_{yx} = \mu\frac{du}{dy} \quad \text{then} \quad \mu\frac{d^2u}{dy^2} = -\rho g \quad \text{and} \quad \frac{d^2u}{dy^2} = -\frac{\rho g}{\mu}$$

Integrating with respect to y gives

$$\frac{du}{dy} = -\frac{\rho g}{\mu}y + c_1$$

Integrating again, we obtain

$$u = -\frac{\rho g}{\mu}\frac{y^2}{2} + c_1 y + c_2$$

To evaluate constants c_1 and c_2, we apply appropriate boundary conditions:

(i) $y = 0, \quad u = 0$ (no-slip)

(ii) $y = \delta, \quad \dfrac{du}{dy} = 0$ (neglect air resistance, i.e., assume zero shear stress at free surface)

From boundary condition (i), $c_2 = 0$

From boundary condition (ii), $0 = -\frac{\rho g}{\mu}\delta + c_1$ or $c_1 = \frac{\rho g}{\mu}\delta$

Hence,

$$u = -\frac{\rho g}{\mu}\frac{y^2}{2} + \frac{\rho g}{\mu}\delta y \quad \text{or} \quad u = \frac{\rho g}{\mu}\delta^2\left[\left(\frac{y}{\delta}\right) - \frac{1}{2}\left(\frac{y}{\delta}\right)^2\right] \longleftarrow u(y)$$

Using the velocity profile it can be shown that:

the volume flow rate is $Q/l = \frac{\rho g}{3\mu}\delta^3$

the maximum velocity is $U_{\max} = \frac{\rho g}{2\mu}\delta^2$

the average velocity is $\overline{V} = \frac{\rho g}{3\mu}\delta^2$

Flow in the liquid film is laminar for $Re = \overline{V}\delta/\nu \leq 1000$ [1].

Notes:
✓ This problem is a special case ($\theta = 90°$) of the inclined plate flow analyzed in Example 5.9 that we solved using the Navier–Stokes equations.
✓ This problem and Example 5.9 demonstrate that use of the differential control volume approach or the Navier–Stokes equations leads to the same result.

8.3 Fully Developed Laminar Flow in a Pipe

VIDEO

Laminar Flow Exiting from a Tube.

As a final example of fully developed laminar flow, let us consider fully developed laminar flow in a pipe. Here the flow is axisymmetric. Consequently it is most convenient to work in cylindrical coordinates. This is yet another case where we could use the Navier–Stokes equations, this time in cylindrical coordinates (Eqs. B.3). Instead we will again take the longer route—using a differential control volume—to bring out some important features of the fluid mechanics. The development will be very similar to that for parallel plates in the previous section; cylindrical coordinates just make the analysis a little trickier mathematically. Since the flow is axisymmetric, the control volume will be a differential annulus, as shown in Fig. 8.7. The control volume length is dx and its thickness is dr.

For a fully developed steady flow, the x component of the momentum equation (Eq. 4.18a), when applied to the differential control volume, once again reduces to

$$F_{S_x} = 0$$

The next step is to sum the forces acting on the control volume in the x direction. We know that normal forces (pressure forces) act on the left and right ends of the control volume, and that tangential forces (shear forces) act on the inner and outer cylindrical surfaces.

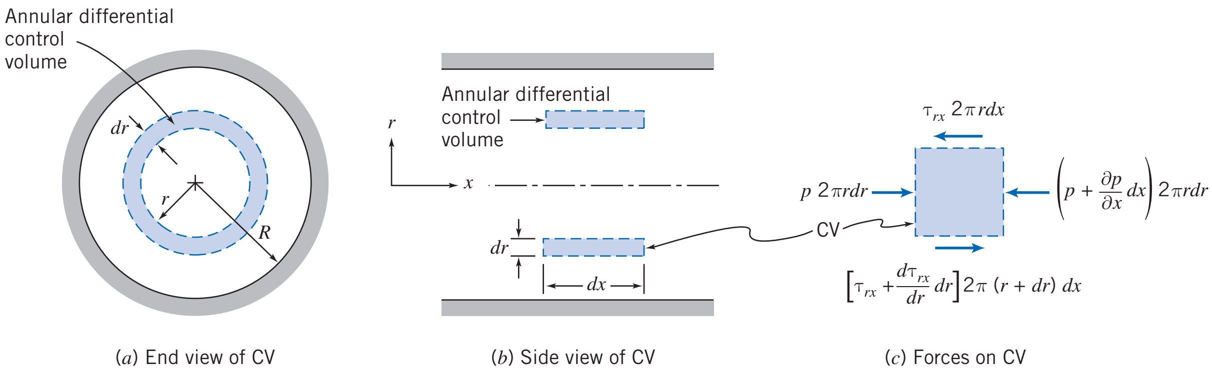


Fig. 8.7 Differential control volume for analysis of fully developed laminar flow in a pipe.

If the pressure at the left face of the control volume is p, then the pressure force on the left end is

$$dF_L = p2\pi r\,dr$$

The pressure force on the right end is

$$dF_R = -\left(p + \frac{\partial p}{\partial x}\,dx\right)2\pi r\,dr$$

If the shear stress at the inner surface of the annular control volume is τ_{rx}, then the shear force on the inner cylindrical surface is

$$dF_I = -\tau_{rx}2\pi r\,dx$$

The shear force on the outer cylindrical surface is

$$dF_O = \left(\tau_{rx} + \frac{d\tau_{rx}}{dr}\,dr\right)2\pi\,(r + dr)dx$$

The sum of the x components of force, dF_L, dF_R, dF_I, and dF_O, acting on the control volume must be zero. This leads to the condition that

$$-\frac{\partial p}{\partial x}\,2\pi r\,dr\,dx + \tau_{rx}2\pi\,dr\,dx + \frac{d\tau_{rx}}{dr}\,2\pi r\,dr\,dx = 0$$

Dividing this equation by $2\pi r\,dr\,dx$ and solving for $\partial p/\partial x$ gives

$$\frac{\partial p}{\partial x} = \frac{\tau_{rx}}{r} + \frac{d\tau_{rx}}{dr} = \frac{1}{r}\,\frac{d(r\tau_{rx})}{dr}$$

Comparing this to the corresponding equation for parallel plates (Eq. 8.3) shows the mathematical complexity introduced because we have cylindrical coordinates. The left side of the equation is at most a function of x only (the pressure is uniform at each section); the right side is at most a function of r only (because the flow is fully developed). Hence, the only way the equation can be valid for all x and r is for both sides to in fact be constant:

$$\frac{1}{r}\,\frac{d(r\tau_{rx})}{dr} = \frac{\partial p}{\partial x} = \text{constant} \quad \text{or} \quad \frac{d(r\tau_{rx})}{dr} = r\frac{\partial p}{\partial x}$$

We are not quite finished, but already we have an important result: *In a constant diameter pipe, the pressure drops uniformly along the pipe length* (except for the entrance region).

Integrating this equation, we obtain

$$r\tau_{rx} = \frac{r^2}{2}\left(\frac{\partial p}{\partial x}\right) + c_1$$

or

$$\tau_{rx} = \frac{r}{2}\left(\frac{\partial p}{\partial x}\right) + \frac{c_1}{r} \tag{8.10}$$

Since $\tau_{rx} = \mu du/dr$, we have

$$\mu\frac{du}{dr} = \frac{r}{2}\left(\frac{\partial p}{\partial x}\right) + \frac{c_1}{r}$$

and

$$u = \frac{r^2}{4\mu}\left(\frac{\partial p}{\partial x}\right) + \frac{c_1}{\mu}\ln r + c_2 \tag{8.11}$$

We need to evaluate constants c_1 and c_2. However, we have only the one boundary condition that $u = 0$ at $r = R$. What do we do? Before throwing in the towel, let us look at the solution for the velocity profile given by Eq. 8.11. Although we do not know the velocity at the pipe centerline, we do know from physical considerations that the velocity must be finite at $r = 0$. The only way that this can be true is for c_1 to be zero. (We could have also concluded that $c_1 = 0$ from Eq. 8.10—which would otherwise yield an infinite stress at $r = 0$.) Thus, from physical considerations, we conclude that $c_1 = 0$, and hence

$$u = \frac{r^2}{4\mu}\left(\frac{\partial p}{\partial x}\right) + c_2$$

The constant, c_2, is evaluated by using the available boundary condition at the pipe wall: at $r = R$, $u = 0$. Consequently,

$$0 = \frac{R^2}{4\mu}\left(\frac{\partial p}{\partial x}\right) + c_2$$

This gives

$$c_2 = -\frac{R^2}{4\mu}\left(\frac{\partial p}{\partial x}\right)$$

and hence

$$u = \frac{r^2}{4\mu}\left(\frac{\partial p}{\partial x}\right) - \frac{R^2}{4\mu}\left(\frac{\partial p}{\partial x}\right) = \frac{1}{4\mu}\left(\frac{\partial p}{\partial x}\right)(r^2 - R^2)$$

or

$$u = -\frac{R^2}{4\mu}\left(\frac{\partial p}{\partial x}\right)\left[1 - \left(\frac{r}{R}\right)^2\right] \tag{8.12}$$

Since we have the velocity profile, we can obtain a number of additional features of the flow.

Shear Stress Distribution

The shear stress is

$$\tau_{rx} = \mu\frac{du}{dr} = \frac{r}{2}\left(\frac{\partial p}{\partial x}\right) \tag{8.13a}$$

Volume Flow Rate

The volume flow rate is

$$\begin{aligned} Q &= \int_A \vec{V}\cdot d\vec{A} = \int_0^R u2\pi r\,dr = \int_0^R \frac{1}{4\mu}\left(\frac{\partial p}{\partial x}\right)(r^2 - R^2)2\pi r\,dr \\ Q &= -\frac{\pi R^4}{8\mu}\left(\frac{\partial p}{\partial x}\right) \end{aligned} \tag{8.13b}$$

Flow Rate as a Function of Pressure Drop

We proved that in fully developed flow the pressure gradient, $\partial p/\partial x$, is constant. Therefore, $\partial p/\partial x = (p_2 - p_1)/L = -\Delta p/L$. Substituting into Eq. 8.13b for the volume flow rate gives

$$Q = -\frac{\pi R^4}{8\mu}\left[\frac{-\Delta p}{L}\right] = \frac{\pi \Delta p R^4}{8\mu L} = \frac{\pi \Delta p D^4}{128\mu L} \qquad (8.13c)$$

for laminar flow in a horizontal pipe. Note that Q is a sensitive function of D; $Q \sim D^4$, so, for example, doubling the diameter D increases the flow rate Q by a factor of 16.

Average Velocity

The average velocity magnitude, $\overline{V}$, is given by

$$\overline{V} = \frac{Q}{A} = \frac{Q}{\pi R^2} = -\frac{R^2}{8\mu}\left(\frac{\partial p}{\partial x}\right) \qquad (8.13d)$$

Point of Maximum Velocity

To find the point of maximum velocity, we set du/dr equal to zero and solve for the corresponding r. From Eq. 8.12

$$\frac{du}{dr} = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right) r$$

Thus,

$$\frac{du}{dr} = 0 \quad \text{at} \quad r = 0$$

At $r = 0$,

$$u = u_{\max} = U = -\frac{R^2}{4\mu}\left(\frac{\partial p}{\partial x}\right) = 2\overline{V} \qquad (8.13e)$$

The velocity profile (Eq. 8.12) may be written in terms of the maximum (centerline) velocity as

$$\frac{u}{U} = 1 - \left(\frac{r}{R}\right)^2 \qquad (8.14)$$

The parabolic velocity profile, given by Eq. 8.14 for fully developed laminar pipe flow, was sketched in Fig. 8.1.

Example 8.4 CAPILLARY VISCOMETER

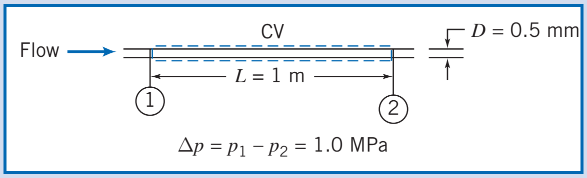
A simple and accurate viscometer can be made from a length of capillary tubing. If the flow rate and pressure drop are measured, and the tube geometry is known, the viscosity of a Newtonian liquid can be computed from Eq. 8.13c. A test of a certain liquid in a capillary viscometer gave the following data:

Flow rate:	880 mm^3/s	Tube length:	1 m
Tube diameter:	0.50 mm	Pressure drop:	1.0 MPa

Determine the viscosity of the liquid.

Given: Flow in a capillary viscometer.
The flow rate is $Q = 880\ \text{mm}^3/\text{s}$.

Find: The fluid viscosity.



Solution:

Equation 8.13c may be applied.

Governing equation:

$$Q = \frac{\pi \Delta p D^4}{128 \mu L} \tag{8.13c}$$

Assumptions:
(1) Laminar flow.
(2) Steady flow.
(3) Incompressible flow.
(4) Fully developed flow.
(5) Horizontal tube.

Then

$$\mu = \frac{\pi \Delta p D^4}{128\, LQ} = \frac{\pi}{128} \times 1.0 \times 10^6 \frac{\text{N}}{\text{m}^2} \times (0.50)^4\ \text{mm}^4 \times \frac{\text{s}}{880\ \text{mm}^3} \times \frac{1}{1\ \text{m}} \times \frac{\text{m}}{10^3\ \text{mm}}$$

$$\mu = 1.74 \times 10^{-3}\ \text{N} \cdot \text{s/m}^2 \longleftarrow \mu$$

Check the Reynolds number. Assume the fluid density is similar to that of water, 999 kg/m³. Then

$$\overline{V} = \frac{Q}{A} = \frac{4Q}{\pi D^2} = \frac{4}{\pi} \times 880 \frac{\text{mm}^3}{\text{s}} \times \frac{1}{(0.50)^2\ \text{mm}^2} \times \frac{\text{m}}{10^3\ \text{mm}} = 4.48\ \text{m/s}$$

and

$$Re = \frac{\rho \overline{V} D}{\mu} = 999 \frac{\text{kg}}{\text{m}^3} \times 4.48 \frac{\text{m}}{\text{s}} \times 0.50\ \text{mm} \times \frac{\text{m}^2}{1.74 \times 10^{-3}\ \text{N} \cdot \text{s}} \times \frac{\text{m}}{10^3\ \text{mm}} \times \frac{\text{N} \cdot \text{s}^2}{\text{kg} \cdot \text{m}}$$

$$Re = 1290$$

Consequently, since $Re < 2300$, the flow is laminar.

This problem is a little oversimplified. To design a capillary viscometer the entrance length, liquid temperature, and kinetic energy of the flowing liquid would all need to be considered.

Part B Flow in Pipes and Ducts

In this section we will be interested in determining the factors that affect the pressure in an incompressible fluid as it flows in a pipe or duct (we will refer to "pipe" but imply "duct," too). If we ignore friction for a moment (and assume steady flow and consider a streamline in the flow), the Bernoulli equation from Chapter 6 applies,

$$\frac{p}{\rho} + \frac{V^2}{2} + gz = \text{constant} \tag{6.8}$$

From this equation we can see what *tends* to lead to a *pressure decrease* along the streamline in this frictionless flow: a *reduction of area* at some point in the pipe (causing an increase in the velocity V), or the pipe having a *positive incline* (so z increases). Conversely, the pressure will tend to increase if the flow area is increased or the pipe slopes downward. We say "tends to" because one factor may counteract another; for example, we may have a downward sloping pipe (tending to increase pressure) with a reduction in diameter (tending to decrease pressure).

In reality, flows in pipes and ducts experience significant friction and are often turbulent, so the Bernoulli equation does not apply (it doesn't even make sense to use V; instead we will use $\overline{V}$, to represent the average velocity at a section along the pipe). We will learn that, in effect, friction effects lead to a continual reduction in the value of the Bernoulli constant of Eq. 6.8 (this represents a "loss" of mechanical energy). We have already seen that, in contrast to the Bernoulli equation, for laminar flow there is a pressure drop even for a horizontal, constant diameter pipe; in this section we will see that turbulent flows experience an even larger pressure drop. We will need to replace the Bernoulli equation with an energy equation that incorporates the effects of friction.

In summary, we can state that *three* factors tend to reduce the pressure in a pipe flow: a decrease in pipe area, an upward slope, and friction. For now we will focus on pressure loss due to friction and so will analyze pipes that are of constant area and that are horizontal.

We have already seen in the previous section that for laminar flow we can theoretically deduce the pressure drop. Rearranging Eq. 8.13c to solve for the pressure drop Δp,

$$\Delta p = \frac{128\mu LQ}{\pi D^4}$$

We would like to develop a similar expression that applies for turbulent flows, but we will see that this is not possible analytically; instead, we will develop expressions based on a combination of theoretical and experimental approaches. Before proceeding, we note that it is conventional to break losses due to friction into two categories: *major losses*, which are losses due to friction in the constant-area sections of the pipe; and *minor losses* (sometimes larger than "major" losses), which are losses due to valves, elbows, and so on (and we will treat the pressure drop at the entrance region as a minor loss term).

Since circular pipes are most common in engineering applications, the basic analysis will be performed for circular geometries. The results can be extended to other geometries by introducing the hydraulic diameter, which is treated in Section 8.7. (Open channel flows will be treated in Chapter 11, and compressible flow in ducts will be treated in Chapter 13.)

8.4 Shear Stress Distribution in Fully Developed Pipe Flow

We consider again fully developed flow in a horizontal circular pipe, except now we may have laminar or turbulent flow. In Section 8.3 we showed that a force balance between friction and pressure forces leads to Eq. 8.10:

$$\tau_{rx} = \frac{r}{2}\left(\frac{\partial p}{\partial x}\right) + \frac{c_1}{r} \tag{8.10}$$

Because we cannot have infinite stress at the centerline, the constant of integration c_1 must be zero, so

$$\tau_{rx} = \frac{r}{2}\frac{\partial p}{\partial x} \tag{8.15}$$

Equation 8.15 indicates that *for both laminar and turbulent fully developed flows the shear stress varies linearly across the pipe,* from zero at the centerline to a maximum at

the pipe wall. The stress on the wall, τ_w (equal and opposite to the stress in the fluid at the wall), is given by

$$\tau_w = -\left[\tau_{rx}\right]_{r=R} = -\frac{R}{2}\frac{\partial p}{\partial x} \tag{8.16}$$

For *laminar* flow we used our familiar stress equation $\tau_{rx} = \mu\, du/dr$ in Eq. 8.15 to eventually obtain the laminar velocity distribution. This led to a set of usable equations, Eqs. 8.13, for obtaining various flow characteristics; e.g., Eq. 8.13c gave a relationship for the flow rate Q, a result first obtained experimentally by Jean Louis Poiseuille, a French physician, and independently by Gotthilf H. L. Hagen, a German engineer, in the 1850s [2].

Unfortunately there is no equivalent stress equation for *turbulent* flow, so we cannot replicate the laminar flow analysis to derive turbulent equivalents of Eqs. 8.13. All we can do in this section is indicate some classic semi-empirical results [3].

As we discussed in Section 2.6, and illustrated in Fig. 2.17, turbulent flow is represented at each point by the time-mean velocity $\overline{u}$ plus (for a two-dimensional flow) randomly fluctuating velocity components u' and v' in the x and y directions (in this context y is the distance inwards from the pipe wall). These components continuously transfer momentum between adjacent fluid layers, tending to reduce any velocity gradient present. This effect shows up as an apparent stress, first introduced by Osborne Reynolds, and called the *Reynolds stress*.[1] This stress is given by $-\rho\overline{u'v'}$, where the overbar indicates a time average. Hence, we find

$$\tau = \tau_{\text{lam}} + \tau_{\text{turb}} = \mu\frac{d\overline{u}}{dy} - \rho\overline{u'v'} \tag{8.17}$$

Do not misunderstand the minus sign in Eq. 8.17—it turns out that u' and v' are negatively correlated, so that $\tau_{\text{turb}} = -\rho\overline{u'v'}$ is positive. In Fig. 8.8, experimental measurements of the Reynolds stress for fully developed turbulent pipe flow at two Reynolds numbers are presented; $Re_U = UD/\nu$, where U is the centerline velocity. The turbulent shear stress has been nondimensionalized with the wall shear stress. Recall that Eq. 8.15 showed that the shear stress in the fluid varies linearly from τ_w at the pipe wall ($y/R \to 0$) to zero at the centerline ($y/R = 1$); from Fig. 8.8 we see that the Reynolds stress has almost the same trend, so that the friction is almost all due to Reynolds stress. What Fig. 8.8 *doesn't* show is that close to the wall ($y/R \to 0$) the Reynolds stress drops to zero. This is because the no-slip condition holds, so not

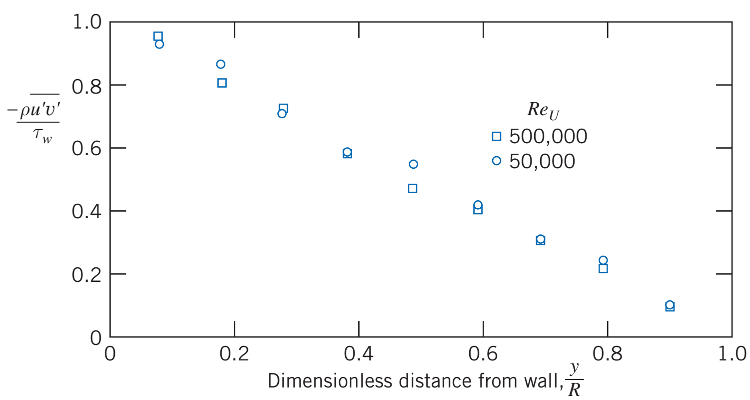


Fig. 8.8 Turbulent shear stress (Reynolds stress) for fully developed turbulent flow in a pipe. (Data from Laufer [5].)

[1]The Reynolds stress terms arise from consideration of the complete equations of motion for turbulent flow [4].

only does the mean velocity $\overline{u} \to 0$, but also the fluctuating velocity components u' and $v' \to 0$ (the wall tends to suppress the fluctuations). Hence, the turbulent stress, $\tau_{\text{turb}} = -\rho\overline{u'v'} \to 0$, as we approach the wall, and is zero at the wall. Since the Reynolds stress is zero at the wall, Eq. 8.17 shows that the wall shear is given by $\tau_w = \mu(d\overline{u}/dy)_{y=0}$. In the region very close to the wall, called the *wall layer*, viscous shear is dominant. In the region between the wall layer and the central portion of the pipe both viscous and turbulent shear are important.

8.5 Turbulent Velocity Profiles in Fully Developed Pipe Flow

Except for flows of very viscous fluids in small diameter ducts, internal flows generally are turbulent. As noted in the discussion of shear stress distribution in fully developed pipe flow (Section 8.4), in turbulent flow there is no universal relationship between the stress field and the mean velocity field. Thus, for turbulent flows we are forced to rely on experimental data.

Dividing Eq. 8.17 by ρ gives

$$\frac{\tau}{\rho} = \nu\frac{d\overline{u}}{dy} - \overline{u'v'} \tag{8.18}$$

The term τ/ρ arises frequently in the consideration of turbulent flows; it has dimensions of velocity squared. In particular, the quantity $(\tau_w/\rho)^{1/2}$ is called the *friction velocity* and is denoted by the symbol u_*. It is a constant for a given flow.

The velocity profile for fully developed turbulent flow through a smooth pipe is shown in Fig. 8.9. The plot is semilogarithmic; $\overline{u}/u_*$ is plotted against $\log(yu_*/\nu)$. The nondimensional parameters $\overline{u}/u_*$ and yu_*/ν arise from dimensional analysis if one reasons that the velocity in the neighborhood of the wall is determined by the conditions at the wall, the fluid properties, and the distance from the wall. It is simply fortuitous that the dimensionless plot of Fig. 8.9 gives a fairly accurate representation of the velocity profile in a pipe away from the wall; note the small deviations in the region of the pipe centerline.

VIDEO

The Glen Canyon Dam: A Turbulent Pipe Flow.

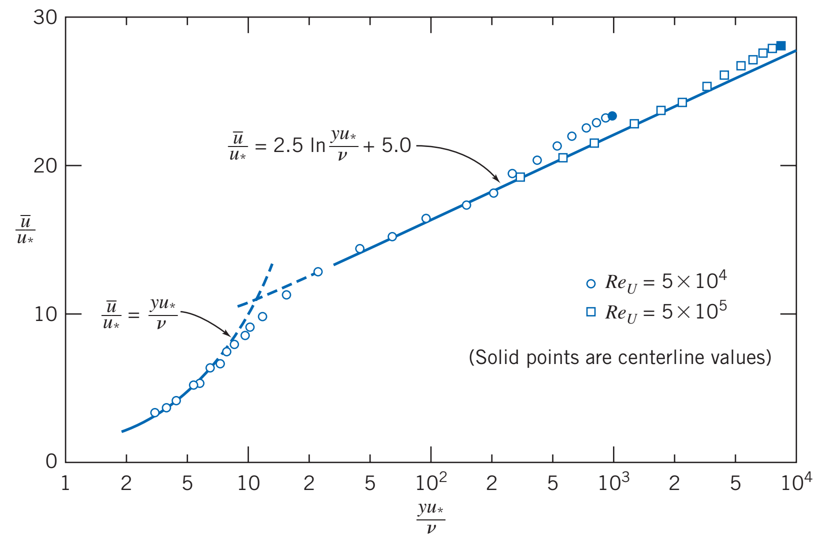
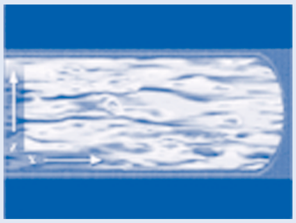


Fig. 8.9 Turbulent velocity profile for fully developed flow in a smooth pipe. (Data from Laufer [5].)

VIDEO

Computer Simulation: Turbulent Channel Flow 1.



VIDEO

Computer Simulation: Turbulent Channel Flow 2.

VIDEO

Computer Simulation: Turbulent Channel Flow 3.

In the region very close to the wall where viscous shear is dominant, the mean velocity profile follows the linear viscous relation

$$u^+ = \frac{\bar{u}}{u_*} = \frac{yu_*}{\nu} = y^+ \tag{8.19}$$

where y is distance measured from the wall ($y = R - r$; R is the pipe radius), and $\bar{u}$ is mean velocity. Equation 8.19 is valid for $0 \leq y^+ \leq 5\text{–}7$; this region is called the *viscous sublayer*.

For values of $yu_*/\nu > 30$, the data are quite well represented by the semilogarithmic curve-fit equation

$$\frac{\bar{u}}{u_*} = 2.5 \ln \frac{yu_*}{\nu} + 5.0 \tag{8.20}$$

In this region both viscous and turbulent shear are important (although turbulent shear is expected to be significantly larger). There is considerable scatter in the numerical constants of Eq. 8.20; the values given represent averages over many experiments [6]. The region between $y^+ = 5\text{–}7$ and $y^+ = 30$ is referred to as the *transition region*, or *buffer layer*.

If Eq. 8.20 is evaluated at the centerline ($y = R$ and $u = U$) and the general expression of Eq. 8.20 is subtracted from the equation evaluated at the centerline, we obtain

$$\frac{U - \bar{u}}{u_*} = 2.5 \ln \frac{R}{y} \tag{8.21}$$

where U is the centerline velocity. Equation 8.21, referred to as the *defect law*, shows that the velocity defect (and hence the general shape of the velocity profile in the neighborhood of the centerline) is a function of the distance ratio only and does not depend on the viscosity of the fluid.

The velocity profile for turbulent flow through a smooth pipe may also be approximated by the empirical *power-law* equation

$$\frac{\bar{u}}{U} = \left(\frac{y}{R}\right)^{1/n} = \left(1 - \frac{r}{R}\right)^{1/n} \tag{8.22}$$

where the exponent, n, varies with the Reynolds number. In Fig. 8.10 the data of Laufer [5] are shown on a plot of $\ln y/R$ versus $\ln \bar{u}/U$. If the power-law profile were an accurate representation of the data, all data points would fall on a straight line of slope n. Clearly the data for $Re_U = 5 \times 10^4$ deviate from the best-fit straight line in the neighborhood of the wall.

Hence the power-law profile is not applicable close to the wall ($y/R < 0.04$). Since the velocity is low in this region, the error in calculating integral quantities such as mass, momentum, and energy fluxes at a section is relatively small. The power-law profile gives an infinite velocity gradient at the wall and hence cannot be used in calculations of wall shear stress. Although the profile fits the data close to the centerline, it fails to give zero slope there. Despite these shortcomings, the power-law profile is found to give adequate results in many calculations.

Data from Hinze [7] suggest that the variation of power-law exponent n with Reynolds number (based on pipe diameter, D, and centerline velocity, U) for fully developed flow in smooth pipes is given by

$$n = -1.7 + 1.8 \log Re_U \tag{8.23}$$

for $Re_U > 2 \times 10^4$.

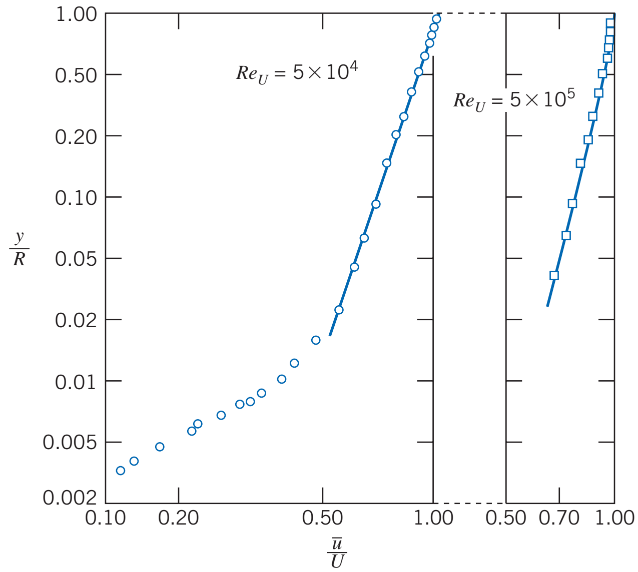


Fig. 8.10 Power-law velocity profiles for fully developed turbulent flow in a smooth pipe. (Data from Laufer [5].)

Since the average velocity is $\overline{V} = Q/A$, and

$$Q = \int_A \vec{V} \cdot d\vec{A}$$

the ratio of the average velocity to the centerline velocity may be calculated for the power-law profiles of Eq. 8.22 assuming the profiles to be valid from wall to centerline. The result is

$$\frac{\overline{V}}{U} = \frac{2n^2}{(n+1)(2n+1)} \tag{8.24}$$

From Eq. 8.24, we see that as n increases (with increasing Reynolds number) the ratio of the average velocity to the centerline velocity increases; with increasing Reynolds number the velocity profile becomes more blunt or "fuller" (for $n = 6$, $\overline{V}/U = 0.79$ and for $n = 10$, $\overline{V}/U = 0.87$). As a representative value, 7 often is used for the exponent; this gives rise to the term "a one-seventh power profile" for fully developed turbulent flow:

$$\frac{\overline{u}}{U} = \left(\frac{y}{R}\right)^{1/7} = \left(1 - \frac{r}{R}\right)^{1/7}$$

Velocity profiles for $n = 6$ and $n = 10$ are shown in Fig. 8.11. The parabolic profile for fully developed laminar flow is included for comparison. It is clear that the turbulent profile has a much steeper slope near the wall. This is consistent with our discussion leading to Eq. 8.17—the fluctuating velocity components u' and v' continuously transfer momentum between adjacent fluid layers, tending to reduce the velocity gradient.

Energy Considerations in Pipe Flow 8.6

We have so far used the momentum and conservation of mass equations, in control volume form, to discuss viscous flow. It is obvious that viscous effects will have an

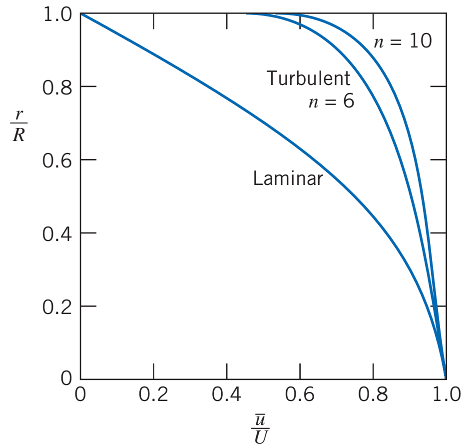


Fig. 8.11 Velocity profiles for fully developed pipe flow.

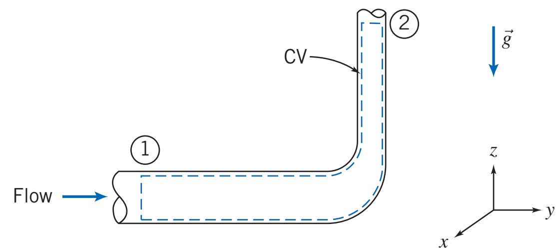


Fig. 8.12 Control volume and coordinates for energy analysis of flow through a 90° reducing elbow.

important effect on energy considerations. In Section 6.5 we discussed the Energy Grade Line (EGL),

$$EGL = \frac{p}{\rho g} + \frac{V^2}{2g} + z \tag{6.16b}$$

and saw that this is a measure of the total mechanical energy ("pressure," kinetic and potential, per unit mass) in a flow. We can expect that instead of being constant (which it was for inviscid flow), the EGL will continuously decrease in the direction of flow as friction "eats" the mechanical energy (Examples 8.9 and 8.10 have sketches of such EGL—and HGL—curves; you may wish to preview them). We can now consider the energy equation (the first law of thermodynamics) to obtain information on the effects of friction.

Consider, for example, steady flow through the piping system, including a reducing elbow, shown in Fig. 8.12. The control volume boundaries are shown as dashed lines. They are normal to the flow at sections ① and ② and coincide with the inside surface of the pipe wall elsewhere.

Basic equation:

$$\dot{Q} - \cancelto{0(1)}{\dot{W}_s} - \cancelto{0(2)}{\dot{W}_{\text{shear}}} - \cancelto{0(1)}{\dot{W}_{\text{other}}} = \cancelto{0(3)}{\frac{\partial}{\partial t}\int_{\text{CV}} e\,\rho\, d\forall} + \int_{\text{CS}} (e + pv)\rho \vec{V} \cdot d\vec{A} \tag{4.56}$$

$$e = u + \frac{V^2}{2} + gz$$

Assumptions: (1) $\dot{W}_s = 0,\ \dot{W}_{\text{other}} = 0$.
(2) $\dot{W}_{\text{shear}} = 0$ (although shear stresses are present at the walls of the elbow, the velocities are zero there, so there is no possibility of work).
(3) Steady flow.
(4) Incompressible flow.
(5) Internal energy and pressure uniform across sections ① and ②.

Under these assumptions the energy equation reduces to

$$\dot{Q} = \dot{m}(u_2 - u_1) + \dot{m}\left(\frac{p_2}{\rho} - \frac{p_1}{\rho}\right) + \dot{m}g(z_2 - z_1) + \int_{A_2} \frac{V_2^2}{2}\rho V_2\, dA_2 - \int_{A_1} \frac{V_1^2}{2}\rho V_1\, dA_1 \tag{8.25}$$

Note that we have *not* assumed the velocity to be uniform at sections ① and ②, since we know that for viscous flows the velocity at a cross-section cannot be uniform. However, it is convenient to introduce the average velocity into Eq. 8.25 so that we can eliminate the integrals. To do this, we define a kinetic energy coefficient.

Kinetic Energy Coefficient

The *kinetic energy coefficient*, α, is defined such that

$$\int_A \frac{V^2}{2}\rho V\, dA = \alpha \int_A \frac{\overline{V}^2}{2}\rho V dA = \alpha \dot{m}\frac{\overline{V}^2}{2} \tag{8.26a}$$

or

$$\alpha = \frac{\displaystyle\int_A \rho V^3 dA}{\dot{m}\overline{V}^2} \tag{8.26b}$$

We can think of α as a correction factor that allows us to use the average velocity $\overline{V}$ in the energy equation to compute the kinetic energy at a cross section.

For laminar flow in a pipe (velocity profile given by Eq. 8.12), $\alpha = 2.0$.

In turbulent pipe flow, the velocity profile is quite flat, as shown in Fig. 8.11. We can use Eq. 8.26b together with Eqs. 8.22 and 8.24 to determine α. Substituting the power-law velocity profile of Eq. 8.22 into Eq. 8.26b, we obtain

$$\alpha = \left(\frac{U}{\overline{V}}\right)^3 \frac{2n^2}{(3+n)(3+2n)} \tag{8.27}$$

Equation 8.24 gives $\overline{V}/U$ as a function of the power-law exponent n; combining this with Eq. 8.27 leads to a fairly complicated expression in n. The overall result is that in the realistic range of n, from $n = 6$ to $n = 10$ for high Reynolds numbers, α varies from 1.08 to 1.03; for the one-seventh power profile ($n = 7$), $\alpha = 1.06$. Because α is reasonably close to unity for high Reynolds numbers, and because the change in kinetic energy is usually small compared with the dominant terms in the energy equation, *we shall almost always use the approximation* $\alpha = 1$ *in our pipe flow calculations*.

Head Loss

Using the definition of α, the energy equation (Eq. 8.25) can be written

$$\dot{Q} = \dot{m}(u_2 - u_1) + \dot{m}\left(\frac{p_2}{\rho} - \frac{p_1}{\rho}\right) + \dot{m}g(z_2 - z_1) + \dot{m}\left(\frac{\alpha_2\overline{V}_2^2}{2} - \frac{\alpha_1\overline{V}_1^2}{2}\right)$$

Dividing by the mass flow rate gives

$$\frac{\delta Q}{dm} = u_2 - u_1 + \frac{p_2}{\rho} - \frac{p_1}{\rho} + gz_2 - gz_1 + \frac{\alpha_2 \overline{V}_2^2}{2} - \frac{\alpha_1 \overline{V}_1^2}{2}$$

Rearranging this equation, we write

$$\left(\frac{p_1}{\rho} + \alpha_1 \frac{\overline{V}_1^2}{2} + gz_1\right) - \left(\frac{p_2}{\rho} + \alpha_2 \frac{\overline{V}_2^2}{2} + gz_2\right) = (u_2 - u_1) - \frac{\delta Q}{dm} \tag{8.28}$$

In Eq. 8.28, the term

$$\left(\frac{p}{\rho} + \alpha \frac{\overline{V}^2}{2} + gz\right)$$

represents the mechanical energy per unit mass at a cross section. (Compare it to the EGL expression, Eq. 6.16b, for computing "mechanical" energy, which we discussed at the beginning of this section. The differences are that in the EGL we divide by g to obtain the EGL in units of feet or meters, and here $\alpha\overline{V}^2$ allows for the fact that in a pipe flow we have a velocity profile, not a uniform flow.) The term $u_2 - u_1 - \delta Q/dm$ is equal to the difference in mechanical energy per unit mass between sections ① and ②. It represents the (irreversible) conversion of mechanical energy at section ① to unwanted thermal energy $(u_2 - u_1)$ and loss of energy via heat transfer $(-\delta Q/dm)$. We identify this group of terms as the total energy loss per unit mass and designate it by the symbol h_{l_T}. Then

$$\left(\frac{p_1}{\rho} + \alpha_1 \frac{\overline{V}_1^2}{2} + gz_1\right) - \left(\frac{p_2}{\rho} + \alpha_2 \frac{\overline{V}_2^2}{2} + gz_2\right) = h_{l_T} \tag{8.29}$$

The dimensions of energy per unit mass FL/M are equivalent to dimensions of L^2/t^2. Equation 8.29 is one of the most important and useful equations in fluid mechanics. It enables us to compute the loss of mechanical energy caused by friction between two sections of a pipe. We recall our discussion at the beginning of Part B, where we discussed what would cause the pressure to change. We hypothesized a frictionless flow (i.e., described by the Bernoulli equation, or Eq. 8.29 with $\alpha = 1$ and $h_{l_T} = 0$) so that the pressure could only change if the velocity changed (if the pipe had a change in diameter), or if the potential changed (if the pipe was not horizontal). Now, with friction, Eq. 8.29 indicates that the pressure will change even for a constant-area horizontal pipe—mechanical energy will be continuously changed into thermal energy.

As the empirical science of hydraulics developed during the 19th century, it was common practice to express the energy balance in terms of energy per unit *weight* of flowing liquid (e.g., water) rather than energy per unit *mass*, as in Eq. 8.29. When Eq. 8.29 is divided by the acceleration of gravity, g, we obtain

$$\left(\frac{p_1}{\rho g} + \alpha_1 \frac{\overline{V}_1^2}{2g} + z_1\right) - \left(\frac{p_2}{\rho g} + \alpha_2 \frac{\overline{V}_2^2}{2g} + z_2\right) = \frac{h_{l_T}}{g} = H_{l_T} \tag{8.30}$$

Each term in Eq. 8.30 has dimensions of energy per unit weight of flowing fluid. Then the net dimensions of $H_{l_T} = h_{l_T}/g$ are $(L^2/t^2)(t^2/L) = L$, or feet of flowing liquid. Since the term head loss is in common use, we shall use it when referring to either H_{l_T} (with dimensions of energy per unit weight or length) or $h_{l_T} = gH_{l_T}$ (with dimensions of energy per unit mass).

Equation 8.29 (or Eq. 8.30) can be used to calculate the pressure difference between any two points in a piping system, provided the head loss, h_{l_T} (or H_{l_T}), can be determined. We shall consider calculation of head loss in the next section.

Calculation of Head Loss 8.7

Total head loss, h_{l_T}, is regarded as the sum of major losses, h_l, due to frictional effects in fully developed flow in constant-area tubes, and minor losses, h_{l_m}, resulting from entrances, fittings, area changes, and so on. Consequently, we consider the major and minor losses separately.

Major Losses: Friction Factor

The energy balance, expressed by Eq. 8.29, can be used to evaluate the major head loss. For fully developed flow through a constant-area pipe, $h_{l_m} = 0$, and $\alpha_1 (\overline{V}_1^2/2) = \alpha_2 (\overline{V}_2^2/2)$; Eq. 8.29 reduces to

$$\frac{p_1 - p_2}{\rho} = g(z_2 - z_1) + h_l \tag{8.31}$$

If the pipe is horizontal, then $z_2 = z_1$ and

$$\frac{p_1 - p_2}{\rho} = \frac{\Delta p}{\rho} = h_l \tag{8.32}$$

Thus the major head loss can be expressed as the pressure loss for fully developed flow through a horizontal pipe of constant area.

Since head loss represents the energy converted by frictional effects from mechanical to thermal energy, head loss for fully developed flow in a constant-area duct depends only on the details of the flow through the duct. Head loss is independent of pipe orientation.

a. Laminar Flow

In laminar flow, we saw in Section 8.3 that the pressure drop may be computed analytically for fully developed flow in a horizontal pipe. Thus, from Eq. 8.13c,

$$\Delta p = \frac{128\mu L Q}{\pi D^4} = \frac{128\mu L \overline{V}(\pi D^2/4)}{\pi D^4} = 32\,\frac{L}{D}\,\frac{\mu \overline{V}}{D}$$

Substituting in Eq. 8.32 gives

$$h_l = 32\frac{L}{D}\frac{\mu \overline{V}}{\rho D} = \frac{L}{D}\frac{\overline{V}^2}{2}\left(64\frac{\mu}{\rho \overline{V} D}\right) = \left(\frac{64}{Re}\right)\frac{L}{D}\frac{\overline{V}^2}{2} \tag{8.33}$$

(We shall see the reason for writing h_l in this form shortly.)

b. Turbulent Flow

In turbulent flow we cannot evaluate the pressure drop analytically; we must resort to experimental results and use dimensional analysis to correlate the experimental data. In fully developed turbulent flow, the pressure drop, Δp, caused by friction in a horizontal constant-area pipe is known to depend on pipe diameter, D, pipe length, L, pipe roughness, e, average flow velocity, $\overline{V}$, fluid density, ρ, and fluid viscosity, μ. In functional form

$$\Delta p = \Delta p(D, L, e, \overline{V}, \rho, \mu)$$

We applied dimensional analysis to this problem in Example 7.2. The results were a correlation of the form

$$\frac{\Delta p}{\rho \overline{V}^2} = f\left(\frac{\mu}{\rho \overline{V} D}, \frac{L}{D}, \frac{e}{D}\right)$$

We recognize that $\mu/\rho \overline{V} D = 1/Re$, so we could just as well write

$$\frac{\Delta p}{\rho \overline{V}^2} = \phi\left(Re, \frac{L}{D}, \frac{e}{D}\right)$$

Substituting from Eq. 8.32, we see that

$$\frac{h_l}{\overline{V}^2} = \phi\left(Re, \frac{L}{D}, \frac{e}{D}\right)$$

Although dimensional analysis predicts the functional relationship, we must obtain actual values experimentally.

Experiments show that the nondimensional head loss is directly proportional to L/D. Hence we can write

$$\frac{h_l}{\overline{V}^2} = \frac{L}{D}\,\phi_1\left(Re, \frac{e}{D}\right)$$

Since the function, ϕ_1, is still undetermined, it is permissible to introduce a constant into the left side of the above equation. By convention the number $\frac{1}{2}$ is introduced into the denominator so that the left side of the equation is the ratio of the head loss to the kinetic energy per unit mass of flow. Then

$$\frac{h_l}{\frac{1}{2}\overline{V}^2} = \frac{L}{D}\phi_2\left(Re, \frac{e}{D}\right)$$

The unknown function, $\phi_2(Re,\ e/D)$, is defined as the *friction factor*, *f*,

$$f \equiv \phi_2\left(Re, \frac{e}{D}\right)$$

and

$$h_l = f\,\frac{L}{D}\,\frac{\overline{V}^2}{2} \tag{8.34}$$

or

$$H_l = f\frac{L}{D}\frac{\overline{V}^2}{2g} \tag{8.35}$$

The friction factor[2] is determined experimentally. The results, published by L. F. Moody [8], are shown in Fig. 8.13.

To determine head loss for fully developed flow with known conditions, the Reynolds number is evaluated first. Roughness, e, is obtained from Table 8.1. Then the friction factor, f, can be read from the appropriate curve in Fig. 8.13, at the known values of Re and e/D. Finally, head loss can be found using Eq. 8.34 or Eq. 8.35.

[2]The friction factor defined by Eq. 8.34 is the *Darcy friction factor*. The *Fanning friction factor*, less frequently used, is defined in Problem 8.95.

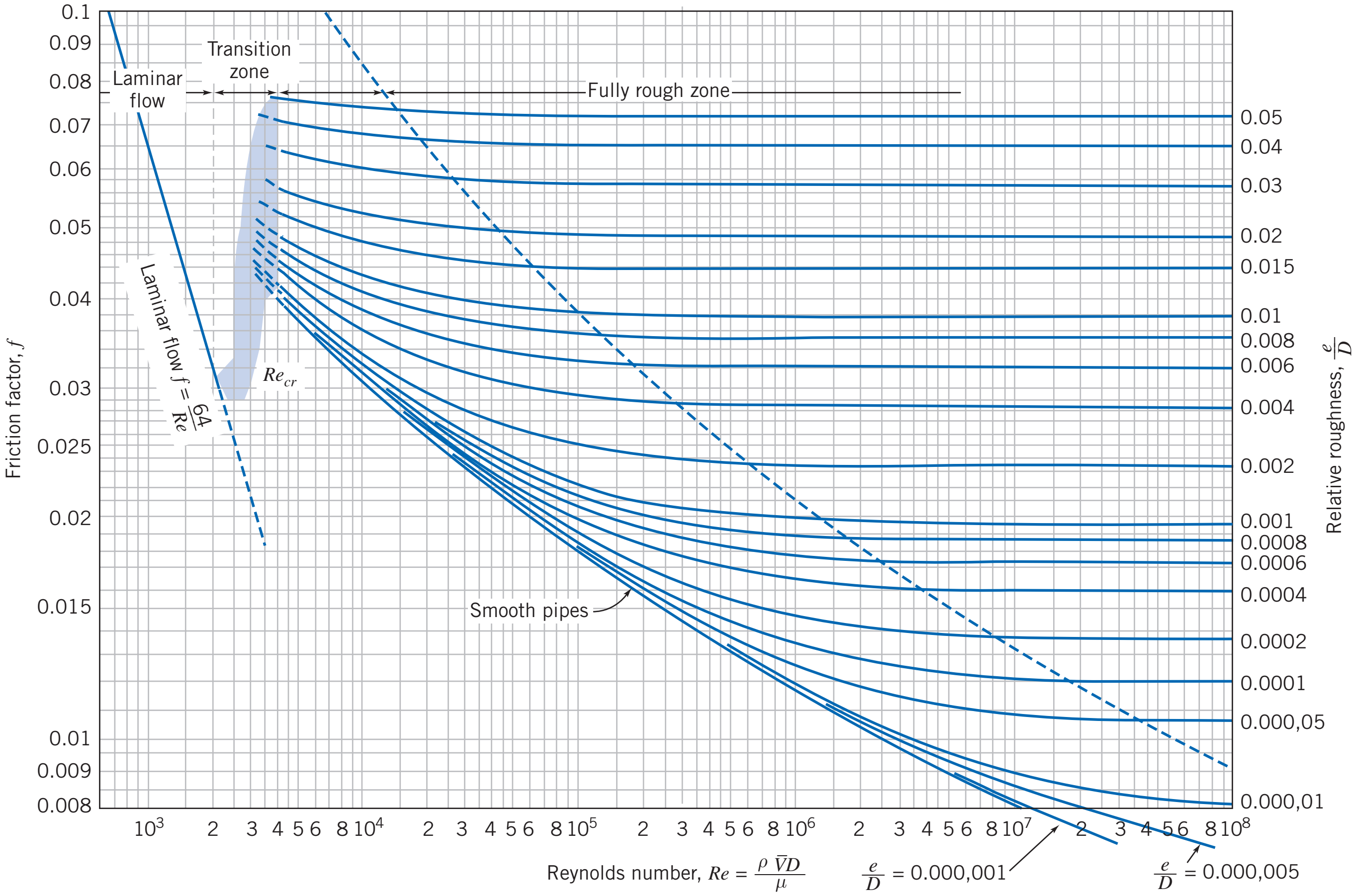


Fig. 8.13 Friction factor for fully developed flow in circular pipes. (Data from Moody [8], used by permission.)

Table 8.1

Roughness for Pipes of Common Engineering Materials

	Roughness, e	
Pipe	Feet	Millimeters
Riveted steel	0.003–0.03	0.9–9
Concrete	0.001–0.01	0.3–3
Wood stave	0.0006–0.003	0.2–0.9
Cast iron	0.00085	0.26
Galvanized iron	0.0005	0.15
Asphalted cast iron	0.0004	0.12
Commercial steel or wrought iron	0.00015	0.046
Drawn tubing	0.000005	0.0015

Source: Data from Moody [8].

Several features of Fig. 8.13 require some discussion. The friction factor for laminar flow may be obtained by comparing Eqs. 8.33 and 8.34:

$$h_l = \left(\frac{64}{Re}\right)\frac{L}{D}\frac{\bar{V}^2}{2} = f\,\frac{L}{D}\frac{\bar{V}^2}{2}$$

Consequently, for laminar flow

$$f_{\text{laminar}} = \frac{64}{Re} \tag{8.36}$$

Thus, in laminar flow, the friction factor is a function of Reynolds number only; it is independent of roughness. Although we took no notice of roughness in deriving Eq. 8.33, experimental results verify that the friction factor is a function only of Reynolds number in laminar flow.

The Reynolds number in a pipe may be changed most easily by varying the average flow velocity. If the flow in a pipe is originally laminar, increasing the velocity until the critical Reynolds number is reached causes transition to occur; the laminar flow gives way to turbulent flow. The effect of transition on the velocity profile was discussed in Section 8.5. Figure 8.11 shows that the velocity gradient at the tube wall is much larger for turbulent flow than for laminar flow. This change in velocity profile causes the wall shear stress to increase sharply, with the same effect on the friction factor.

As the Reynolds number is increased above the transition value, the velocity profile continues to become fuller, as noted in Section 8.5. For values of relative roughness $e/D \leq 0.001$, the friction factor at first tends to follow the smooth pipe curve, along which friction factor is a function of Reynolds number only. However, as the Reynolds number increases, the velocity profile becomes still fuller. The size of the thin viscous sublayer near the tube wall decreases. As roughness elements begin to poke through this layer, the effect of roughness becomes important, and the friction factor becomes a function of both the Reynolds number *and* the relative roughness.

At very large Reynolds number, most of the roughness elements on the tube wall protrude through the viscous sublayer; the drag and, hence, the pressure loss, depend only on the size of the roughness elements. This is termed the "fully rough" flow regime; the friction factor depends only on e/D in this regime.

For values of relative roughness $e/D \geq 0.001$, as the Reynolds number is increased above the transition value, the friction factor is greater than the smooth pipe value. As was the case for lower values of e/D, the value of Reynolds number at which the flow regime becomes fully rough decreases with increasing relative roughness.

To summarize the preceding discussion, we see that as Reynolds number is increased, the friction factor decreases as long as the flow remains laminar. At transition, f increases sharply. In the turbulent flow regime, the friction factor decreases gradually and finally levels out at a constant value for large Reynolds number.

Bear in mind that the actual loss of energy is h_l (Eq. 8.34), which is proportional to f and $\overline{V}^2$. Hence, for laminar flow $h_l \propto \overline{V}$ (because $f = 64/Re$, and $Re \propto \overline{V}$); for the transition region there is a sudden increase in h_l; for the fully rough zone $h_l \propto \overline{V}^2$ (because $f \approx$ const.), and for the rest of the turbulent region h_l increases at a rate somewhere between $\overline{V}$ and $\overline{V}^2$. We conclude that the head loss *always* increases with flow rate, and more rapidly when the flow is turbulent.

To avoid having to use a graphical method for obtaining f for turbulent flows, various mathematical expressions have been fitted to the data. The most widely used formula for friction factor is from Colebrook [9],

$$\frac{1}{\sqrt{f}} = -2.0 \log\left(\frac{e/D}{3.7} + \frac{2.51}{Re\sqrt{f}}\right) \tag{8.37}$$

Equation 8.37 is implicit in f, but these days most scientific calculators have an equation-solving feature that can be easily used to find f for a given roughness ratio e/D and Reynolds number Re (and some calculators have the Colebrook equation itself built in!). Certainly a spreadsheet such as *Excel*, or other mathematical computer

applications, can also be used (there is an *Excel* add-in for computing f for laminar and turbulent flows available on the Web site). Even without using these automated approaches, Eq. 8.37 is not difficult to solve for f—all we need to do is iterate. Equation 8.37 is quite stable—almost any initial guess value for f in the right side will, after very few iterations, lead to a converged value for f to three significant figures. From Fig. 8.13, we can see that for turbulent flows $f < 0.1$; hence $f = 0.1$ would make a good initial value. Another strategy is to use Fig. 8.13 to obtain a good first estimate; then usually one iteration using Eq. 8.37 yields a good value for f. As an alternative, Haaland [10] developed the following equation,

$$\frac{1}{\sqrt{f}} = -1.8 \log\left[\left(\frac{e/D}{3.7}\right)^{1.11} + \frac{6.9}{Re}\right]$$

as an approximation to the Colebrook equation; for $Re > 3000$, it gives results within about 2 percent of the Colebrook equation, without the need to iterate.

For turbulent flow in smooth pipes, the Blasius correlation, valid for $Re \leq 10^5$, is

$$f = \frac{0.316}{Re^{0.25}} \tag{8.38}$$

When this relation is combined with the expression for wall shear stress (Eq. 8.16), the expression for head loss (Eq. 8.32), and the definition of friction factor (Eq. 8.34), a useful expression for the wall shear stress is obtained as

$$\tau_w = 0.0332\rho\overline{V}^2\left(\frac{\nu}{R\overline{V}}\right)^{0.25} \tag{8.39}$$

This equation will be used later in our study of turbulent boundary-layer flow over a flat plate (Chapter 9).

All of the e values given in Table 8.1 are for new pipes, in relatively good condition. Over long periods of service, corrosion takes place and, particularly in hard water areas, lime deposits and rust scale form on pipe walls. Corrosion can weaken pipes, eventually leading to failure. Deposit formation increases wall roughness appreciably, and also decreases the effective diameter. These factors combine to cause e/D to increase by factors of 5 to 10 for old pipes (see Problem 10.63). An example is shown in Fig. 8.14.

Curves presented in Fig. 8.13 represent average values for data obtained from numerous experiments. The curves should be considered accurate within approximately ± 10 percent, which is sufficient for many engineering analyses. If more accuracy is needed, actual test data should be used.

Minor Losses

The flow in a piping system may be required to pass through a variety of fittings, bends, or abrupt changes in area. Additional head losses are encountered, primarily as a result of flow separation. (Energy eventually is dissipated by violent mixing in the separated zones.) These losses will be minor (hence the term *minor losses*) if the piping system includes long lengths of constant-area pipe. Depending on the device, minor losses traditionally are computed in one of two ways, either

$$h_{l_m} = K\frac{\overline{V}^2}{2} \tag{8.40a}$$

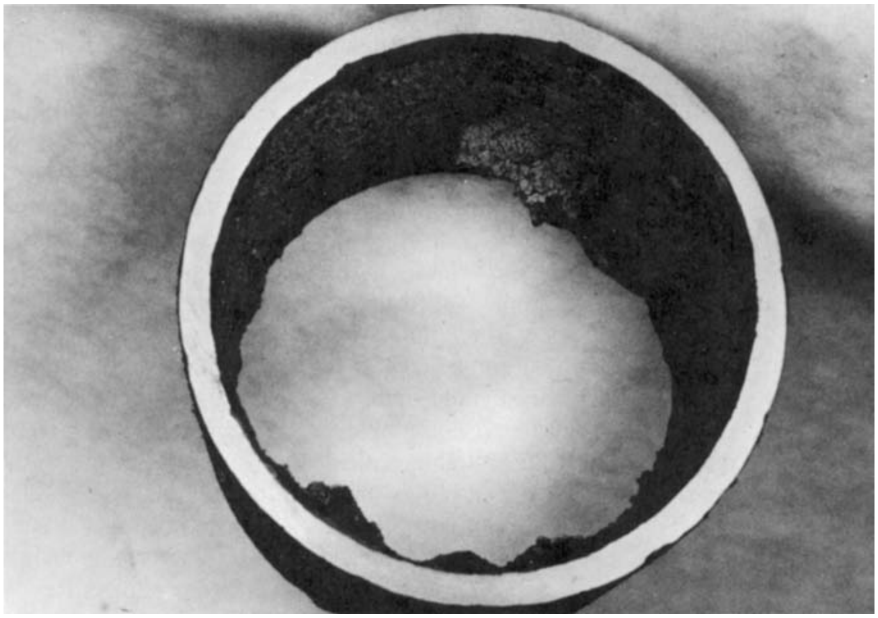


Fig. 8.14 Pipe section removed after 40 years of service as a water line, showing formation of scale. (Photo courtesy of Alan T. McDonald.)

where the *loss coefficient*, K, must be determined experimentally for each situation, or

$$h_{l_m} = f\frac{L_e}{D}\frac{\overline{V}^2}{2} \tag{8.40b}$$

where L_e is an *equivalent length* of straight pipe.

For flow through pipe bends and fittings, the loss coefficient, K, is found to vary with pipe size (diameter) in much the same manner as the friction factor, f, for flow through a straight pipe. Consequently, the equivalent length, L_e/D, tends toward a constant for different sizes of a given type of fitting.

Experimental data for minor losses are plentiful, but they are scattered among a variety of sources. Different sources may give different values for the same flow configuration. The data presented here should be considered as representative for some commonly encountered situations; in each case the source of the data is identified.

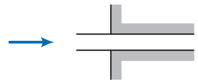
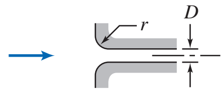
a. Inlets and Exits

A poorly designed inlet to a pipe can cause appreciable head loss. If the inlet has sharp corners, flow separation occurs at the corners, and a *vena contracta* is formed. The fluid must accelerate locally to pass through the reduced flow area at the vena contracta. Losses in mechanical energy result from the unconfined mixing as the flow stream decelerates again to fill the pipe. Three basic inlet geometries are shown in Table 8.2. From the table it is clear that the loss coefficient is reduced significantly when the inlet is rounded even slightly. For a well-rounded inlet ($r/D \geq 0.15$) the entrance loss coefficient is almost negligible. Example 8.9 illustrates a procedure for experimentally determining the loss coefficient for a pipe inlet.

The kinetic energy per unit mass, $\alpha\overline{V}^2/2$, is completely dissipated by mixing when flow discharges from a duct into a large reservoir or plenum chamber. The situation corresponds to flow through an abrupt expansion with $AR = 0$ (Fig. 8.15). The minor loss coefficient thus equals α, which as we saw in the previous section we usually set

Table 8.2

Minor Loss Coefficients for Pipe Entrances

Entrance Type		Minor Loss Coefficient, K^a
Reentrant		0.78
Square-edged		0.5
Rounded		r/D: 0.02, 0.06, ≥ 0.15; K: 0.28, 0.15, 0.04

r/D	0.02	0.06	≥ 0.15
K	0.28	0.15	0.04

[a]Based on $h_{l_m} = K(\bar{V}^2/2)$, where $\bar{V}$ is the mean velocity in the pipe.

Source: Data from Reference [11].

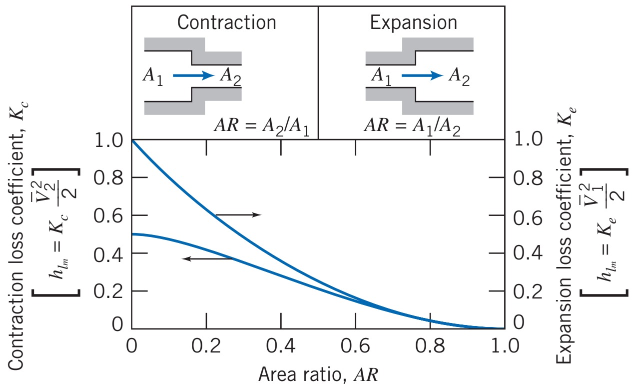


Fig. 8.15 Loss coefficients for flow through sudden area changes. (Data from Streeter [1].)

to 1 for turbulent flow. No improvement in minor loss coefficient for an exit is possible; however, addition of a diffuser can reduce $\bar{V}^2/2$ and therefore h_{l_m} considerably (see Example 8.10).

b. Enlargements and Contractions

Minor loss coefficients for sudden expansions and contractions in circular ducts are given in Fig. 8.15. Note that both loss coefficients are based on the *larger* $\bar{V}^2/2$. Thus losses for a sudden expansion are based on $\bar{V}_1^2/2$, and those for a contraction are based on $\bar{V}_2^2/2$.

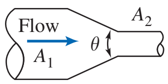
Losses caused by area change can be reduced somewhat by installing a nozzle or diffuser between the two sections of straight pipe. Data for nozzles are given in Table 8.3. Note that the final column (data for the included angle $\theta = 180°$) agrees with the data of Fig. 8.15.

Losses in diffusers depend on a number of geometric and flow variables. Diffuser data most commonly are presented in terms of a pressure recovery coefficient, C_p, defined as the ratio of static pressure rise to inlet dynamic pressure,

$$C_p \equiv \frac{p_2 - p_1}{\frac{1}{2}\rho \bar{V}_1^2} \qquad (8.41)$$

Table 8.3

Loss Coefficients (*K*) for Gradual Contractions: Round and Rectangular Ducts

		Included Angle, θ, Degrees						
	A_2/A_1	10	15–40	50–60	90	120	150	180
	0.50	0.05	0.05	0.06	0.12	0.18	0.24	0.26
	0.25	0.05	0.04	0.07	0.17	0.27	0.35	0.41
	0.10	0.05	0.05	0.08	0.19	0.29	0.37	0.43

Note: Coefficients are based on $h_{l_m} = K(\bar{V}_2^2/2)$.

Source: Data from ASHRAE [12].

This shows what fraction of the inlet kinetic energy shows up as a pressure rise. It is not difficult to show (using the Bernoulli and continuity equations; see Problem 8.201) that the ideal (frictionless) pressure recovery coefficient is given by

$$C_{p_i} = 1 - \frac{1}{AR^2} \tag{8.42}$$

where AR is the area ratio. Hence, the ideal pressure recovery coefficient is a function only of the area ratio. In reality a diffuser typically has turbulent flow, and the static pressure rise in the direction of flow may cause flow separation from the walls if the diffuser is poorly designed; flow pulsations can even occur. For these reasons the actual C_p will be somewhat less than indicated by Eq. 8.42. For example, data for conical diffusers with fully developed turbulent pipe flow at the inlet are presented in Fig. 8.16 as a function of geometry. Note that more tapered diffusers (small divergence angle ϕ or large dimensionless length N/R_1) are more likely to approach the ideal constant value for C_p. As we make the cone shorter, for a given fixed area ratio we start to see a drop in C_p—we can consider the cone length at which this starts to happen the optimum length (it is the shortest length for which we obtain the maximum coefficient for a given area ratio—closest to that predicted by Eq. 8.42). We can relate C_p to the head loss. If gravity is neglected, and $\alpha_1 = \alpha_2 = 1.0$, the head loss equation, Eq. 8.29, reduces to

$$\left[\frac{p_1}{\rho} + \frac{\bar{V}_1^2}{2}\right] - \left[\frac{p_2}{\rho} + \frac{\bar{V}_2^2}{2}\right] = h_{l_T} = h_{l_m}$$

Thus,

$$h_{l_m} = \frac{\bar{V}_1^2}{2} - \frac{\bar{V}_2^2}{2} - \frac{p_2 - p_1}{\rho}$$

$$h_{l_m} = \frac{\bar{V}_1^2}{2}\left[\left(1 - \frac{\bar{V}_2^2}{\bar{V}_1^2}\right) - \frac{p_2 - p_1}{\frac{1}{2}\rho\bar{V}_1^2}\right] = \frac{\bar{V}_1^2}{2}\left[\left(1 - \frac{\bar{V}_2^2}{\bar{V}_1^2}\right) - C_p\right]$$

From continuity, $A_1\bar{V}_1 = A_2\bar{V}_2$, so

$$h_{l_m} = \frac{\bar{V}_1^2}{2}\left[1 - \left(\frac{A_1}{A_2}\right)^2 - C_p\right]$$

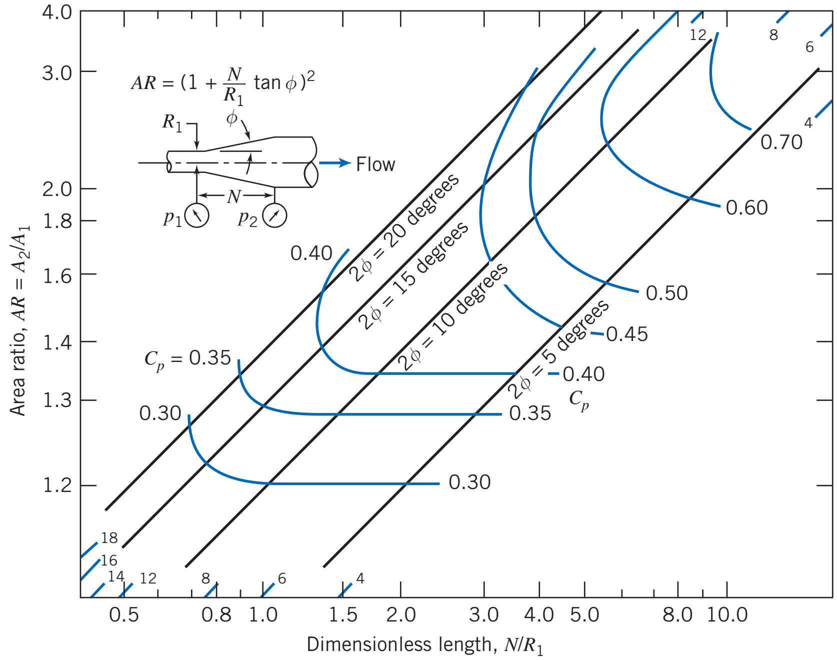


Fig. 8.16 Pressure recovery for conical diffusers with fully developed turbulent pipe flow at inlet. (Data from Cockrell and Bradley [13].)

or

$$h_{l_m} = \frac{\overline{V}_1^2}{2}\left[\left(1 - \frac{1}{(AR)^2}\right) - C_p\right] \tag{8.43}$$

The frictionless result (Eq. 8.42) is obtained from Eq. 8.43 if $h_{l_m} = 0$. We can combine Eqs. 8.42 and 8.43 to obtain an expression for the head loss in terms of the actual and ideal C_p values:

$$h_{l_m} = (C_{p_i} - C_p)\frac{\overline{V}_1^2}{2} \tag{8.44}$$

Performance maps for plane wall and annular diffusers [14] and for radial diffusers [15] are available in the literature.

Diffuser pressure recovery is essentially independent of Reynolds number for inlet Reynolds numbers greater than 7.5×10^4 [16]. Diffuser pressure recovery with uniform inlet flow is somewhat better than that for fully developed inlet flow. Performance maps for plane wall, conical, and annular diffusers for a variety of inlet flow conditions are presented in [17].

Since static pressure rises in the direction of flow in a diffuser, flow may separate from the walls. For some geometries, the outlet flow is distorted. For wide angle diffusers, vanes or splitters can be used to suppress stall and improve pressure recovery [18].

CLASSIC VIDEO

Flow Visualization.

c. Pipe Bends

The head loss of a bend is larger than for fully developed flow through a straight section of equal length. The additional loss is primarily the result of secondary flow, and is represented most conveniently by an equivalent length of straight pipe. The equivalent length depends on the relative radius of curvature of the bend, as shown in

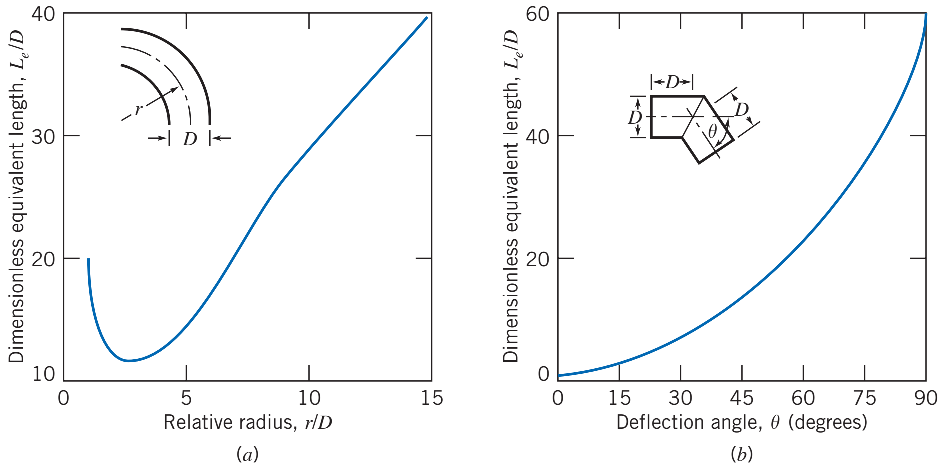


Fig. 8.17 Representative total resistance (L_e/D) for (a) 90° pipe bends and flanged elbows, and (b) miter bends. (Data from Reference [11].)

Table 8.4

Representative Dimensionless Equivalent Lengths (L_e/D) for Valves and Fittings

Fitting Type	Equivalent Length,[a] L_e/D
Valves (fully open)	
Gate valve	8
Globe valve	340
Angle valve	150
Ball valve	3
Lift check valve: globe lift	600
angle lift	55
Foot valve with strainer: poppet disk	420
hinged disk	75
Standard elbow: 90°	30
45°	16
Return bend, close pattern	50
Standard tee: flow through run	20
flow through branch	60

[a]Based on $h_{l_m} = f(L_e/D)(\bar{V}^2/2)$.

Source: Data from Reference [11].

Fig. 8.17*a* for 90° bends. An approximate procedure for computing the resistance of bends with other turning angles is given in [11].

Because they are simple and inexpensive to construct in the field, miter bends often are used in large pipe systems. Design data for miter bends are given in Fig. 8.17*b*. Note that you get what you pay for: From Fig. 8.17a the equivalent length for pipe bends varies from about 10 to about 40 diameters; for the cheaper 90° miter bend of Fig. 8.17b we get a much larger equivalent length of 60 diameters.

d. Valves and Fittings

Losses for flow through valves and fittings also may be expressed in terms of an equivalent length of straight pipe. Some representative data are given in Table 8.4.

All resistances are given for fully open valves; losses increase markedly when valves are partially open. Valve design varies significantly among manufacturers.

Whenever possible, resistances furnished by the valve supplier should be used if accurate results are needed.

Fittings in a piping system may have threaded, flanged, or welded connections. For small diameters, threaded joints are most common; large pipe systems frequently have flanged or welded joints.

In practice, insertion losses for fittings and valves vary considerably, depending on the care used in fabricating the pipe system. If burrs from cutting pipe sections are allowed to remain, they cause local flow obstructions, which increase losses appreciably.

Although the losses discussed in this section were termed "minor losses," they can be a large fraction of the overall system loss. Thus a system for which calculations are to be made must be checked carefully to make sure all losses have been identified and their magnitudes estimated. If calculations are made carefully, the results will be of satisfactory engineering accuracy. You may expect to predict actual losses within ± 10 percent.

We include here one more device that changes the energy of the fluid—except this time the energy of the fluid will be increased, so it creates a "negative energy loss."

Pumps, Fans, and Blowers in Fluid Systems

In many practical flow situations (e.g., the cooling system of an automobile engine, the HVAC system of a building), the driving force for maintaining the flow against friction is a pump (for liquids) or a fan or blower (for gases). Here we will consider pumps, although all the results apply equally to fans and blowers. We generally neglect heat transfer and internal energy changes of the fluid (we will incorporate them later into the definition of the pump efficiency), so the first law of thermodynamics applied across the pump is

$$\dot{W}_{\text{pump}} = \dot{m}\left[\left(\frac{p}{\rho} + \frac{\overline{V}^2}{2} + gz\right)_{\text{discharge}} - \left(\frac{p}{\rho} + \frac{\overline{V}^2}{2} + gz\right)_{\text{suction}}\right]$$

We can also compute the head Δh_{pump} (energy/mass) produced by the pump,

$$\Delta h_{\text{pump}} = \frac{\dot{W}_{\text{pump}}}{\dot{m}} = \left(\frac{p}{\rho} + \frac{\overline{V}^2}{2} + gz\right)_{\text{discharge}} - \left(\frac{p}{\rho} + \frac{\overline{V}^2}{2} + gz\right)_{\text{suction}} \tag{8.45}$$

In many cases the inlet and outlet diameters (and therefore velocities) and elevations are the same or negligibly different, so Eq. 8.45 simplifies to

$$\Delta h_{\text{pump}} = \frac{\Delta p_{\text{pump}}}{\rho} \tag{8.46}$$

It is interesting to note that a pump adds energy to the fluid in the form of a gain in pressure—the everyday, invalid perception is that pumps add kinetic energy to the fluid. (It *is* true that when a pump-pipe system is first started up, the pump does work to accelerate the fluid to its steady speed; this is when a pump driven by an electric motor is most in danger of burning out the motor.)

The idea is that in a pump-pipe system the head produced by the pump (Eq. 8.45 or 8.46) is needed to overcome the head loss for the pipe system. Hence, the flow rate in such a system depends on the pump characteristics and the major and minor losses of the pipe system. We will learn in Chapter 10 that the head produced by a given pump is not constant, but varies with flow rate through the pump, leading to the notion of "matching" a pump to a given system to achieve the desired flow rate.

A useful relation is obtained from Eq. 8.46 if we multiply by $\dot{m} = \rho Q$ (Q is the flow rate) and recall that $\dot{m}\Delta h_{\text{pump}}$ is the power supplied to the fluid,

$$\dot{W}_{\text{pump}} = Q\Delta p_{\text{pump}} \tag{8.47}$$

We can also define the pump efficiency:

$$\eta = \frac{\dot{W}_{\text{pump}}}{\dot{W}_{\text{in}}} \tag{8.48}$$

where $\dot{W}_{\text{pump}}$ is the power reaching the fluid, and $\dot{W}_{\text{in}}$ is the power input (usually electrical) to the pump.

We note that, when applying the energy equation (Eq. 8.29) to a pipe system, we may sometimes choose points 1 and 2 so that a pump is included in the system. For these cases we can simply include the head of the pump as a "negative loss":

$$\left(\frac{p_1}{\rho} + \alpha_1\frac{\bar{V}_1^2}{2} + gz_1\right) - \left(\frac{p_2}{\rho} + \alpha_2\frac{\bar{V}_2^2}{2} + gz_2\right) = h_{l_T} - \Delta h_{\text{pump}} \tag{8.49}$$

Noncircular Ducts

The empirical correlations for pipe flow also may be used for computations involving noncircular ducts, provided their cross sections are not too exaggerated. Thus ducts of square or rectangular cross section may be treated if the ratio of height to width is less than about 3 or 4.

The correlations for turbulent pipe flow are extended for use with noncircular geometries by introducing the *hydraulic diameter*, defined as

$$D_h \equiv \frac{4A}{P} \tag{8.50}$$

in place of the diameter, D. In Eq. 8.50, A is cross-sectional area, and P is *wetted perimeter*, the length of wall in contact with the flowing fluid at any cross-section. The factor 4 is introduced so that the hydraulic diameter will equal the duct diameter for a circular cross section. For a circular duct, $A = \pi D^2/4$ and $P = \pi D$, so that

$$D_h = \frac{4A}{P} = \frac{4\left(\frac{\pi}{4}\right)D^2}{\pi D} = D$$

For a rectangular duct of width b and height h, $A = bh$ and $P = 2(b+h)$, so

$$D_h = \frac{4bh}{2(b+h)}$$

If the *aspect ratio*, ar, is defined as $ar = h/b$, then

$$D_h = \frac{2h}{1+ar}$$

for rectangular ducts. For a square duct, $ar = 1$ and $D_h = h$.

As noted, the hydraulic diameter concept can be applied in the approximate range $\frac{1}{4} < ar < 4$. Under these conditions, the correlations for pipe flow give acceptably accurate results for rectangular ducts. Since such ducts are easy and cheap to fabricate

from sheet metal, they are commonly used in air conditioning, heating, and ventilating applications. Extensive data on losses for air flow are available (e.g., see [12, 19]).

Losses caused by secondary flows increase rapidly for more extreme geometries, so the correlations are not applicable to wide, flat ducts, or to ducts of triangular or other irregular shapes. Experimental data must be used when precise design information is required for specific situations.

Solution of Pipe Flow Problems 8.8

Section 8.7 provides us with a complete scheme for solving many different pipe flow problems. For convenience we collect together the relevant computing equations.

The *energy equation*, relating the conditions at any two points 1 and 2 for a single-path pipe system, is

$$\left(\frac{p_1}{\rho} + \alpha_1 \frac{\overline{V}_1^2}{2} + gz_1\right) - \left(\frac{p_2}{\rho} + \alpha_2 \frac{\overline{V}_2^2}{2} + gz_2\right) = h_{l_T} = \sum h_l + \sum h_{l_m} \tag{8.29}$$

This equation expresses the fact that there will be a loss of mechanical energy ("pressure," kinetic and/or potential) in the pipe. Recall that for turbulent flows $\alpha \approx 1$. Note that by judicious choice of points 1 and 2 we can analyze not only the entire pipe system, but also just a certain section of it that we may be interested in. The *total head loss* is given by the sum of the major and minor losses. (Remember that we can also include "negative losses" for any pumps present between points 1 and 2. The relevant form of the energy equation is then Eq. 8.49.)

Each *major loss* is given by

$$h_l = f \frac{L}{D} \frac{\overline{V}^2}{2} \tag{8.34}$$

where the *friction factor* is obtained from

$$f = \frac{64}{Re} \qquad \text{for laminar flow } (Re < 2300) \tag{8.36}$$

or

$$\frac{1}{\sqrt{f}} = -2.0 \log\left(\frac{e/D}{3.7} + \frac{2.51}{Re\sqrt{f}}\right) \qquad \text{for turbulent flow } (Re \geq 2300) \tag{8.37}$$

and Eqs. 8.36 and 8.37 are presented graphically in the Moody chart (Fig. 8.13).

Each *minor loss* is given either by

$$h_{l_m} = K \frac{\overline{V}^2}{2} \tag{8.40a}$$

where K is the device *loss coefficient*, or

$$h_{l_m} = f \frac{L_e}{D} \frac{\overline{V}^2}{2} \tag{8.40b}$$

where L_e is the additional *equivalent length* of pipe.

We also note that the flow rate Q is related to the average velocity $\overline{V}$ at each pipe cross section by

$$Q = \pi \frac{D^2}{4} \overline{V}$$

We will apply these equations first to single-path systems.

Single-Path Systems

In single-path pipe problems we generally know the system configuration (type of pipe material and hence pipe roughness, the number and type of elbows, valves, and other fittings, etc., and changes of elevation), as well as the fluid (ρ and μ) we will be working with. Although not the only possibilities, usually the goal is to determine one of the following:

(a) The pressure drop Δp, for a given pipe (L and D), and flow rate Q.
(b) The pipe length L, for a given pressure drop Δp, pipe diameter D, and flow rate Q.
(c) The flow rate Q, for a given pipe (L and D), and pressure drop Δp.
(d) The pipe diameter D, for a given pipe length L, pressure drop Δp, and flow rate Q.

Each of these cases often arises in real-world situations. For example, case (a) is a necessary step in selecting the correct size pump to maintain the desired flow rate in a system—the pump must be able to produce the system Δp at the specified flow rate Q. (We will discuss this in more detail in Chapter 10.) Cases (a) and (b) are computationally straightforward; we will see that cases (c) and (d) can be a little tricky to evaluate. We will discuss each case, and present an Example for each. The Examples present solutions as you might do them using a calculator, but there is also an *Excel* workbook for each. (Remember that the Web site has an *Excel* add-in that once installed will automatically compute f from Re and e/D!) The advantage of using a computer application such as a spreadsheet is that we do not have to use either the Moody chart (Fig. 8.13) or solve the implicit Colebrook equation (Eq. 8.37) to obtain turbulent friction factors—the application can find them for us! In addition, as we'll see, cases (c) and (d) involve significant iterative calculations that can be avoided by use of a computer application. Finally, once we have a solution using a computer application, engineering "what-ifs" become easy, e.g., if we double the head produced by a pump, how much will the flow rate in a given system increase?

a. Find Δp for a Given L, D, and Q

These types of problems are quite straightforward—the energy equation (Eq. 8.29) can be solved directly for $\Delta p = (p_1 - p_2)$ in terms of known or computable quantities. The flow rate leads to the Reynolds number (or numbers if there is a diameter change) and hence the friction factor (or factors) for the flow; tabulated data can be used for minor loss coefficients and equivalent lengths. The energy equation can then be used to directly obtain the pressure drop. Example 8.5 illustrates this type of problem.

b. Find L for a Given Δp, D, and Q

These types of problems are also straightforward—the energy equation (Eq. 8.29) can be solved directly for L in terms of known or computable quantities. The flow rate again leads to the Reynolds number and hence the friction factor for the flow. Tabulated data can be used for minor loss coefficients and equivalent lengths. The energy equation can then be rearranged and solved directly for the pipe length. Example 8.6 illustrates this type of problem.

c. Find Q for a Given Δp, L, and D

These types of problems require either manual iteration or use of a computer application such as *Excel*. The unknown flow rate or velocity is needed before the Reynolds

number and hence the friction factor can be found. To manually iterate we first solve the energy equation directly for $\overline{V}$ in terms of known quantities and the unknown friction factor f. To start the iterative process we make a guess for f (a good choice is to take a value from the fully turbulent region of the Moody chart because many practical flows are in this region) and obtain a value for $\overline{V}$. Then we can compute a Reynolds number and hence obtain a new value for f. We repeat the iteration process $f \to \overline{V} \to Re \to f$ until convergence (usually only two or three iterations are necessary). A much quicker procedure is to use a computer application. For example, spreadsheets (such as *Excel*) have built-in solving features for solving one or more algebraic equations for one or more unknowns. Example 8.7 illustrates this type of problem.

d. Find D for a Given Δp, L, and Q

These types of problems arise, for example, when we have designed a pump-pipe system and wish to choose the best pipe diameter—the best being the minimum diameter (for minimum pipe cost) that will deliver the design flow rate. We need to manually iterate, or use a computer application such as *Excel*. The unknown diameter is needed before the Reynolds number and relative roughness, and hence the friction factor, can be found. To manually iterate we could first solve the energy equation directly for D in terms of known quantities and the unknown friction factor f, and then iterate from a starting guess for f in a way similar to case (c) above: $f \to D \to Re$ and $e/D \to f$. In practice this is a little unwieldy, so instead to manually find a solution we make successive guesses for D until the corresponding pressure drop Δp (for the given flow rate Q) computed from the energy equation matches the design Δp. As in case (c) a much quicker procedure is to use a computer application. For example, spreadsheets (such as *Excel*) have built-in solving features for solving one or more algebraic equations for one or more unknowns. Example 8.8 illustrates this type of problem.

In choosing a pipe size, it is logical to work with diameters that are available commercially. Pipe is manufactured in a limited number of standard sizes. Some data for standard pipe sizes are given in Table 8.5. For data on extra strong or double extra strong pipes, consult a handbook, e.g., [11]. Pipe larger than 12 in. nominal diameter is produced in multiples of 2 in. up to a nominal diameter of 36 in. and in multiples of 6 in. for still larger sizes.

Table 8.5

Standard Sizes for Carbon Steel, Alloy Steel, and Stainless Steel Pipe

Nominal Pipe Size (in.)	Inside Diameter (in.)	Nominal Pipe Size (in.)	Inside Diameter (in.)
$\frac{1}{8}$	0.269	$2\frac{1}{2}$	2.469
$\frac{1}{4}$	0.364	3	3.068
$\frac{3}{8}$	0.493	4	4.026
$\frac{1}{2}$	0.622	5	5.047
$\frac{3}{4}$	0.824	6	6.065
1	1.049	8	7.981
$1\frac{1}{2}$	1.610	10	10.020
2	2.067	12	12.000

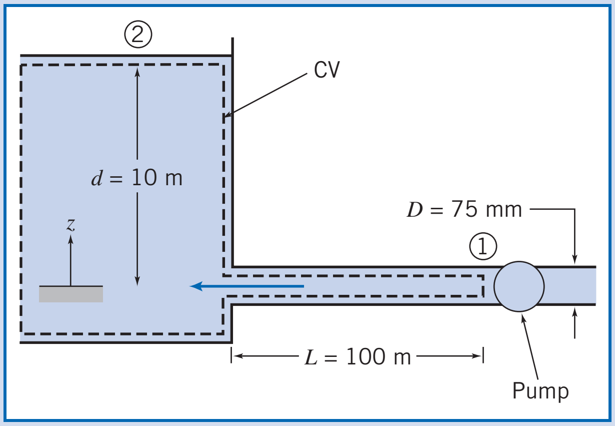
Source: Data from Reference [11].

Example 8.5 PIPE FLOW INTO A RESERVOIR: PRESSURE DROP UNKNOWN

A 100-m length of smooth horizontal pipe is attached to a large reservoir. A pump is attached to the end of the pipe to pump water into the reservoir at a volume flow rate of 0.01 m^3/s. What pressure (gage) must the pump produce at the pipe to generate this flow rate? The inside diameter of the smooth pipe is 75 mm.

Given: Water is pumped at 0.01 m^3/s through a 75-mm-diameter smooth pipe, with $L = 100$ m, into a constant-level reservoir of depth $d = 10$ m.

Find: Pump pressure, p_1, required to maintain the flow.



Solution:

Governing equations:

$$\left(\frac{p_1}{\rho} + \alpha_1 \frac{\overline{V}_1^2}{2} + gz_1\right) - \left(\frac{p_2}{\rho} + \alpha_2 \frac{\overline{V}_2^2}{2} + gz_2\right) = h_{l_T} = h_l + h_{l_m} \tag{8.29}$$

where

$$h_l = f\frac{L}{D}\frac{\overline{V}^2}{2} \quad (8.34) \qquad \text{and} \qquad h_{l_m} = K\frac{\overline{V}^2}{2} \quad (8.40a)$$

For the given problem, $p_1 = p_{\text{pump}}$ and $p_2 = 0$ (gage), so $\Delta p = p_1 - p_2 = p_{\text{pump}}$, $\overline{V}_1 = \overline{V}$, $\overline{V}_2 \approx 0$, K (exit loss) $= 1.0$, and $\alpha_1 \approx 1.0$. If we set $z_1 = 0$, then $z_2 = d$. Simplifying Eq. 8.29 gives

$$\frac{\Delta p}{\rho} + \frac{\overline{V}^2}{2} - gd = f\frac{L}{D}\frac{\overline{V}^2}{2} + \frac{\overline{V}^2}{2} \tag{1}$$

The left side of the equation is the loss of mechanical energy between points 1 and 2; the right side is the major and minor losses that contributed to the loss. Solving for the pressure drop, $\Delta p = p_{\text{pump}}$,

$$p_{\text{pump}} = \Delta p = \rho\left(gd + f\frac{L}{D}\frac{\overline{V}^2}{2}\right)$$

Everything on the right side of the equation is known or can be readily computed. The flow rate Q leads to $\overline{V}$,

$$\overline{V} = \frac{Q}{A} = \frac{4Q}{\pi D^2} = \frac{4}{\pi} \times 0.01\frac{\text{m}^3}{\text{s}} \times \frac{1}{(0.075)^2\,\text{m}^2} = 2.26\,\text{m/s}$$

This in turn [assuming water at 20°C, $\rho = 999$ kg/m^3, and $\mu = 1.0 \times 10^{-3}$ kg/(m·s)] leads to the Reynolds number

$$Re = \frac{\rho\overline{V}D}{\mu} = 999\frac{\text{kg}}{\text{m}^3} \times 2.26\frac{\text{m}}{\text{s}} \times 0.075\,\text{m} \times \frac{\text{m}\cdot\text{s}}{1.0 \times 10^{-3}\text{kg}} = 1.70 \times 10^5$$

For turbulent flow in a smooth pipe ($e = 0$), from Eq. 8.37, $f = 0.0162$. Then

$$p_{\text{pump}} = \Delta p = \rho\left(gd + f\frac{L}{D}\frac{\overline{V}^2}{2}\right)$$

$$= 999\frac{\text{kg}}{\text{m}^3}\left(9.81\frac{\text{m}}{\text{s}^2}\times 10\text{ m} + (0.0162)\times\frac{100\text{ m}}{0.075\text{ m}}\times\frac{(2.26)^2\text{m}^2}{2\text{ s}^2}\right)\times\frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}}$$

$$p_{\text{pump}} = 1.53\times 10^5\ \text{N/m}^2\ \text{(gage)}$$

Hence,

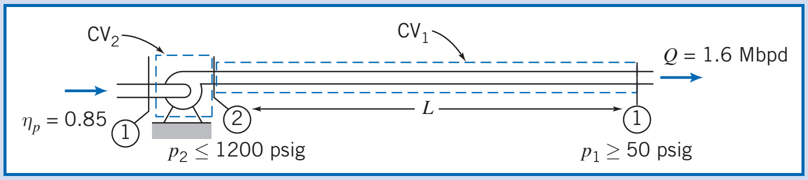
$$p_{\text{pump}} = 153\ \text{kPa (gage)} \longleftarrow p_{\text{pump}}$$

This problem illustrates the method for manually calculating pressure drop. The *Excel* workbook for this problem automatically computes *Re* and *f* from the given data. It then solves Eq. 1 directly for pressure p_{pump} without having to explicitly solve for it first. The workbook can be easily used to see, for example, how the pump pressure p_{pump} required to maintain flow *Q* is affected by changing the diameter *D*; it is easily editable for other case (a) type problems.

Example 8.6 FLOW IN A PIPELINE: LENGTH UNKNOWN

Crude oil flows through a level section of the Alaskan pipeline at a rate of 1.6 million barrels per day (1 barrel = 42 gal). The pipe inside diameter is 48 in.; its roughness is equivalent to galvanized iron. The maximum allowable pressure is 1200 psi; the minimum pressure required to keep dissolved gases in solution in the crude oil is 50 psi. The crude oil has SG = 0.93; its viscosity at the pumping temperature of 140°F is $\mu = 3.5\times 10^{-4}\ \text{lbf}\cdot\text{s/ft}^2$. For these conditions, determine the maximum possible spacing between pumping stations. If the pump efficiency is 85 percent, determine the power that must be supplied at each pumping station.

Given: Flow of crude oil through horizontal section of Alaskan pipeline.



$D = 48$ in. (roughness of galvanized iron), SG = 0.93, $\mu = 3.5\times 10^{-4}\ \text{lbf}\cdot\text{s/ft}^2$

Find: (a) Maximum spacing, *L*.
(b) Power needed at each pump station.

Solution:
As shown in the figure, we assume that the Alaskan pipeline is made up of repeating pump-pipe sections. We can draw two control volumes: CV_1, for the pipe flow (state 2 to state 1); CV_2, for the pump (state 1 to state 2).

First we apply the energy equation for steady, incompressible pipe flow to CV_1.

Governing equations:

$$\left(\frac{p_2}{\rho} + \alpha_2\frac{\cancel{\overline{V}_2^2}}{2} + \cancel{gz_2}\right) - \left(\frac{p_1}{\rho} + \alpha_1\frac{\cancel{\overline{V}_1^2}}{2} + \cancel{gz_1}\right) = h_{l_T} = h_l + h_{l_m} \qquad (8.29)$$

where

$$h_l = f\frac{L}{D}\frac{\overline{V}^2}{2} \quad (8.34) \qquad \text{and} \qquad h_{l_m} = K\frac{\overline{V}^2}{2} \quad (8.40\text{a})$$

Assumptions: (1) $\alpha_1 \overline{V}_1^2 = \alpha_2 \overline{V}_2^2$.
(2) Horizontal pipe, $z_1 = z_2$.
(3) Neglect minor losses.
(4) Constant viscosity.

Then, using CV_1

$$\Delta p = p_2 - p_1 = f \frac{L}{D} \rho \frac{\overline{V}^2}{2} \tag{1}$$

or

$$L = \frac{2D}{f} \frac{\Delta p}{\rho \overline{V}^2} \quad \text{where } f = f(Re, e/D)$$

$$Q = 1.6 \times 10^6 \frac{\text{bbl}}{\text{day}} \times 42 \frac{\text{gal}}{\text{bbl}} \times \frac{\text{ft}^3}{7.48\,\text{gal}} \times \frac{\text{day}}{24\ \text{hr}} \times \frac{\text{hr}}{3600\,\text{s}} = 104\ \text{ft}^3/\text{s}$$

so

$$\overline{V} = \frac{Q}{A} = 104 \frac{\text{ft}^3}{\text{s}} \times \frac{4}{\pi (4)^2 \text{ft}^2} = 8.27\ \text{ft/s}$$

$$Re = \frac{\rho \overline{V} D}{\mu} = (0.93)1.94 \frac{\text{slug}}{\text{ft}^3} \times 8.27 \frac{\text{ft}}{\text{s}} \times 4\,\text{ft} \times \frac{\text{ft}^2}{3.5 \times 10^{-4}\ \text{lbf} \cdot \text{s}} \times \frac{\text{lbf} \cdot \text{s}^2}{\text{slug} \cdot \text{ft}}$$

$$Re = 1.71 \times 10^5$$

From Table 8.1, $e = 0.0005$ ft and hence $e/D = 0.00012$. Then from Eq. 8.37, $f = 0.017$ and thus

$$L = \frac{2}{0.017} \times 4\ \text{ft} \times (1200 - 50) \frac{\text{lbf}}{\text{in.}^2} \times \frac{\text{ft}^3}{(0.93)1.94\ \text{slug}} \times \frac{\text{s}^2}{(8.27)^2\ \text{ft}^2}$$

$$\times 144 \frac{\text{in.}^2}{\text{ft}^2} \times \frac{\text{slug} \cdot \text{ft}}{\text{lbf} \cdot \text{s}^2} = 6.32 \times 10^5 \text{ft}$$

$$L = 632{,}000\text{ft (120 mi)} \longleftarrow L$$

To find the pumping power we can apply the first law of thermodynamics to CV_2. This control volume consists only of the pump, and we saw in Section 8.7 that this law simplifies to

$$\dot{W}_{\text{pump}} = Q \Delta p_{\text{pump}} \tag{8.47}$$

and the pump efficiency is

$$\eta = \frac{\dot{W}_{\text{pump}}}{\dot{W}_{\text{in}}} \tag{8.48}$$

We recall that $\dot{W}_{\text{pump}}$ is the power reaching the fluid, and $\dot{W}_{\text{in}}$ is the power input. Because we have a repeating system the pressure rise through the pump (i.e., from state 1 to state 2) equals the pressure drop in the pipe (i.e., from state 2 to state 1),

$$\Delta p_{\text{pump}} = \Delta p$$

so that

$$\dot{W}_{\text{pump}} = Q\Delta p_{\text{pump}} = 104\frac{\text{ft}^3}{\text{s}} \times \frac{(1200-50)\text{lbf}}{\text{in.}^2} \times \frac{144\ \text{in.}^2}{\text{ft}^2} \times \frac{\text{hp}\cdot\text{s}}{550\text{ft}\cdot\text{lbf}} \approx 31{,}300\ \text{hp}$$

and the required power input is

$$\dot{W}_{\text{in.}} = \frac{\dot{W}_{\text{pump}}}{\eta} = \frac{31300\ \text{hp}}{0.85} = 36{,}800\ \text{hp} \xleftarrow{\qquad\qquad \dot{W}_{\text{needed}}\qquad\qquad}$$

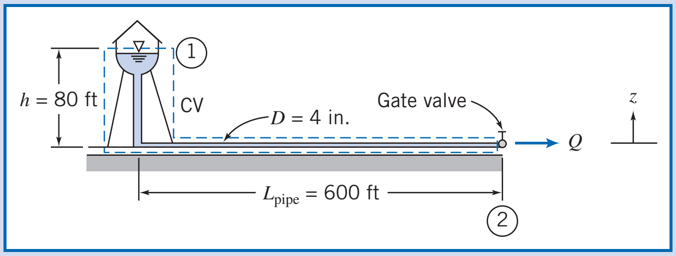
This problem illustrates the method for manually calculating pipe length L. The *Excel* workbook for this problem automatically computes Re and f from the given data. It then solves Eq. 1 directly for L without having to explicitly solve for it first. The workbook can be easily used to see, for example, how the flow rate Q depends on L; it may be edited for other case (b) type problems.

Example 8.7 FLOW FROM A WATER TOWER: FLOW RATE UNKNOWN

A fire protection system is supplied from a water tower and standpipe 80 ft tall. The longest pipe in the system is 600 ft and is made of cast iron about 20 years old. The pipe contains one gate valve; other minor losses may be neglected. The pipe diameter is 4 in. Determine the maximum rate of flow (gpm) through this pipe.

Given: Fire protection system, as shown.

Find: Q, gpm.



Solution:

Governing equations:

$$\left(\frac{\cancel{p_1}}{\rho} + \alpha_1 \overset{\approx 0(2)}{\cancel{\frac{\bar{V}_1^2}{2}}} + gz_1\right) - \left(\frac{\cancel{p_2}}{\rho} + \alpha_2\frac{\bar{V}_2^2}{2} + gz_2\right) = h_{l_T} = h_l + h_{l_m} \qquad (8.29)$$

where

$$h_l = f\frac{L}{D}\frac{\bar{V}^2}{2} \quad (8.34) \qquad \text{and} \qquad h_{l_m} = f\frac{L_e}{D}\frac{\bar{V}^2}{2} \quad (8.40\text{b})$$

Assumptions: (1) $p_1 = p_2 = p_{\text{atm}}$
(2) $\bar{V}_1 = 0$, and $\alpha_2 \simeq 1.0$.

Then Eq. 8.29 can be written as

$$g(z_1 - z_2) - \frac{\bar{V}_2^2}{2} = h_{l_T} = f\left(\frac{L}{D} + \frac{L_e}{D}\right)\frac{\bar{V}_2^2}{2} \qquad (1)$$

For a fully open gate valve, from Table 8.4, $L_e/D = 8$. Thus

$$g(z_1 - z_2) = \frac{\bar{V}_2^2}{2}\left[f\left(\frac{L}{D} + 8\right) + 1\right]$$

To manually iterate, we solve for $\overline{V}_2$ and obtain

$$\overline{V}_2 = \left[\frac{2g(z_1 - z_2)}{f(L/D + 8) + 1}\right]^{1/2} \tag{2}$$

To be conservative, assume the standpipe is the same diameter as the horizontal pipe. Then

$$\frac{L}{D} = \frac{600 \text{ ft} + 80 \text{ ft}}{4 \text{ in.}} \times 12 \frac{\text{in.}}{\text{ft}} = 2040$$

Also

$$z_1 - z_2 = h = 80 \text{ ft}$$

To solve Eq. 2 manually we need to iterate. To start, we make an estimate for f by assuming the flow is fully turbulent (where f is constant). This value can be obtained from solving Eq. 8.37 using a calculator or from Fig. 8.13. For a large value of Re (e.g., 10^8), and a roughness ratio $e/D \approx 0.005$ ($e = 0.00085$ ft for cast iron is obtained from Table 8.1, and doubled to allow for the fact that the pipe is old), we find that $f \approx 0.03$. Thus a first iteration for $\overline{V}_2$ from Eq. 2 is

$$\overline{V}_2 = \left[2 \times 32.2 \frac{\text{ft}}{\text{s}^2} \times 80 \text{ ft} \times \frac{1}{0.03(2040 + 8) + 1}\right]^{1/2} = 9.08 \text{ ft/s}$$

Now obtain a new value for f:

$$Re = \frac{\rho \overline{V} D}{\mu} = \frac{\overline{V} D}{\nu} = 9.08 \frac{\text{ft}}{\text{s}} \times \frac{\text{ft}}{3} \times \frac{\text{s}}{1.21 \times 10^{-5} \text{ ft}^2} = 2.50 \times 10^5$$

For $e/D = 0.005$, $f = 0.0308$ from Eq. 8.37. Thus we obtain

$$\overline{V}_2 = \left[2 \times 32.2 \frac{\text{ft}}{\text{s}^2} \times 80 \text{ ft} \times \frac{1}{0.0308\,(2040 + 8) + 1}\right]^{1/2} = 8.97 \text{ ft/s}$$

The values we have obtained for $\overline{V}_2$ (9.08 ft/s and 8.97 ft/s) differ by less than 2%—an acceptable level of accuracy. If this accuracy had not been achieved we would continue iterating until this, or any other accuracy we desired, was achieved (usually only one or two more iterations at most are necessary for reasonable accuracy). Note that instead of starting with a fully rough value for f, we could have started with a guess value for $\overline{V}_2$ of, say, 1 ft/s or 10 ft/s. The volume flow rate is

$$Q = \overline{V}_2 A = \overline{V}_2 \frac{\pi D^2}{4} = 8.97 \frac{\text{ft}}{s} \times \frac{\pi}{4}\left(\frac{1}{3}\right)^2 \text{ft}^2 \times 7.48 \frac{\text{gal}}{\text{ft}^3} \times 60 \frac{\text{s}}{\text{min}}$$

$$Q = 351 \text{ gpm} \xleftarrow{\qquad\qquad Q \qquad\qquad}$$

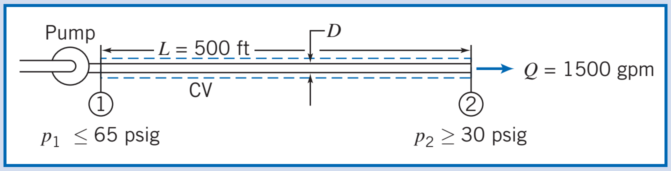
This problem illustrates the method for manually iterating to calculate flow rate.

The *Excel* workbook for this problem automatically iterates to solve for the flow rate Q. It solves Eq. 1 without having to obtain the explicit equation (Eq. 2) for $\overline{V}_2$ (or Q) first. The workbook can be easily used to perform numerous "what-ifs" that would be extremely time-consuming to do manually, e.g., to see how Q is affected by changing the roughness e/D. For example, it shows that replacing the old cast-iron pipe with a new pipe ($e/D \approx 0.0025$) would increase the flow rate from 351 gpm to about 386 gpm, a 10% increase! The workbook can be modified to solve other case (c) type problems.

Example 8.8 FLOW IN AN IRRIGATION SYSTEM: DIAMETER UNKNOWN

Spray heads in an agricultural spraying system are to be supplied with water through 500 ft of drawn aluminum tubing from an engine-driven pump. In its most efficient operating range, the pump output is 1500 gpm at a discharge pressure not exceeding 65 psig. For satisfactory operation, the sprinklers must operate at 30 psig or higher pressure. Minor losses and elevation changes may be neglected. Determine the smallest standard pipe size that can be used.

Given: Water supply system, as shown.



Find: Smallest standard D.

Solution:

Δp, L, and Q are known. D is unknown, so iteration is needed to determine the minimum standard diameter that satisfies the pressure drop constraint at the given flow rate. The maximum allowable pressure drop over the length, L, is

$$\Delta p_{\max} = p_{1_{\max}} - p_{2_{\min}} = (65 - 30)\text{ psi} = 35\text{ psi}$$

Governing equations:

$$\left(\frac{p_1}{\rho} + \alpha_1 \cancel{\frac{\bar{V}_1^2}{2}} + \cancel{g z_1}\right) - \left(\frac{p_2}{\rho} + \alpha_2 \cancel{\frac{\bar{V}_2^2}{2}} + \cancel{g z_2}\right) = h_{l_T} \qquad (8.29)$$

$$= 0(3)$$

$$h_{l_T} = h_l + \cancel{h_{l_m}} = f\frac{L}{D}\frac{\bar{V}_2^2}{2}$$

Assumptions:
(1) Steady flow.
(2) Incompressible flow.
(3) $h_{l_T} = h_l$, i.e., $h_{l_m} = 0$.
(4) $z_1 = z_2$.
(5) $\bar{V}_1 = \bar{V}_2 = \bar{V}$; $\alpha_1 \simeq \alpha_2$.

Then

$$\Delta p = p_1 - p_2 = f\frac{L}{D}\frac{\rho\bar{V}^2}{2} \qquad (1)$$

Equation 1 is difficult to solve for D because both $\bar{V}$ and f depend on D! The best approach is to use a computer application such as *Excel* to automatically solve for D. For completeness here we show the manual iteration procedure. The first step is to express Eq. 1 and the Reynolds number in terms of Q instead of $\bar{V}$ (Q is constant but $\bar{V}$ varies with D). We have $\bar{V} = Q/A = 4Q/\pi D^2$ so that

$$\Delta p = f\frac{L}{D}\frac{\rho}{2}\left(\frac{4Q}{\pi D^2}\right)^2 = \frac{8 f L \rho Q^2}{\pi^2 D^5} \qquad (2)$$

The Reynolds number in terms of Q is

$$Re = \frac{\rho \bar{V} D}{\mu} = \frac{\bar{V} D}{\nu} = \frac{4Q}{\pi D^2}\frac{D}{\nu} = \frac{4Q}{\pi \nu D}$$

Finally, Q must be converted to cubic feet per second.

$$Q = 1500\frac{\text{gal}}{\text{min}} \times \frac{\text{min}}{60\text{ s}} \times \frac{\text{ft}^3}{7.48\text{ gal}} = 3.34\text{ ft}^3/\text{s}$$

For an initial guess, take nominal 4 in. (4.026 in. i.d.) pipe: